THE THEOREMS OF STRUCTURAL AND GEOMETRIC VARIATION FOR LINEAR AND NONLINEAR FINITE ELEMENT ANALYSIS

Abdul Majid ABU KASSIM

A thesis submitted for the degree of

Doctor of Philosophy

Department of Civil Engineering and Building Science

University of Edinburgh

1985

To my parents,

Mariam and Abu Kassim

Page

ABSTRACT

This thesis is concerned with the investigation of a reanalysis technique for finite element problems based on the theorems of structural and geometric variation. A reanalysis technique is normally used to find the response of a modified structure such that the computational effort is less than that required for a fresh analysis.

The theorems of structural variation account for changes in structural properties, whereas the theorems of geometric variation are used for variations in both geometry and structural changes. Both theorems are formulated in matrix form for the efficient reanalysis of linear finite element structures. The efficiency of these theorems are investigated by considering the reanalysis of some simple finite element structures. To emphasize the practical aspects, some potential applications are used to illustrate the efficiency of the theorems for the reanalysis of a sequence of design modification for a structure.

The theorems of geometric variation are then utilized in a series of nonlinear solution techniques based on the Newton-Raphson methods. The proposed techniques are investigated for efficiency by analysing several examples of material and geometrical nonlinearity. These problems indicate which of the proposed techniques are the most efficient as well as the type of problems for which it can be used profitably.

The final part of this thesis is concerned with computer programming. The theorems of structural and geometric variation may readily be coded into existing computer programs without too much difficulty. The task is simpler if the existing programs are highly structured and divided into subprograms. The programs developed were used extensively in this thesis for the comparative tests undertaken.

ACKNOWLEDGEMENTS

The author wishes to thank Professor A.W. Hendry, Head of the Department of Civil Engineering and Building Science, University of Edinburgh, for his support of this research.

The guidance, assistance and valuable suggestions of Dr.B.H.V. Topping during the past three years of this research is gratefully appreciated.

The author is also greatly indebted to the Malaysian Government for their continuing support and funding of this work in the form of a research studentship, without which it would have been impossible.

Many thanks to the advisory staff of ERCC (Edinburgh Regional Computing Centre) for their help and comments in writing the computer programs and using the graphics facilities available. The use of ERCC's word processing and text-editing features of the computer are also gratefully acknowledged.

To the librarian, Mrs.V. Thompson for tracing some rather obscure articles and papers that were invaluable.

To the research students of this department who contributed in the preparation of this thesis in a number of ways.

Last and not least, to the author's parents for their patience and constant encouragement during the past three years.

NOTATIONS

This is a list of principal symbols used in this thesis. Locally used notations and modifications (by addition of superscripts or subscripts) are defined where used. On occasions the same symbol may be used in different contexts and hopefully the appropriate text explanation will avoid confusion.

Mathematical symbols

[ ]	A rectangular or square matrix
$\lceil\ \rfloor$	A diagonal matrix
{ }	A column vector and to save space the components are sometime written in a row
$[\ ]^{-1}$	An inverse matrix
$[\ ]^{T}$	Matrix transpose also applies to vectors
$\left[\begin{array}{c\|c} & \\ \hline & \end{array}\right]$	A partitioned matrix, dotted lines indicate partitioning
d{ }	Differential changes or variations of a column vector also applies to scalars
∂\{ \}	Partial differential of a vector also applies to scalars
$\int$	Integration

Latin symbols

a	Affected nodes of modified elements
A	Area
{b}	Body forces
B	Semi-bandwidth of a matrix
[B]	Strain-displacement matrix
$[B_o]$	Linear strain-displacement matrix
$[B_L]$	Nonlinear strain-displacement matrix
C	Compensation force
CST	Constant strain triangular element
[C]	Matrix of compensation forces
d_f	Displacement factor
{d}	Vector of edge extensions

$[D]$	Elasticity or stress-strain matrix
$[D_{ep}]$	Elasto-plastic matrix
$[D_p]$	Plastic part of elasto-plastic matrix
e	An element, normally used as a superscript
$\{e\}$	Vector of edge strains
E	Elastic or Young's modulus
E_T	Tangent modulus
f	Internal force due to unit load
$\{f\}$	Vector of internal forces due to unit load
$[f]$	Matrix of internal forces due to unit loads or force sensitivity coefficient matrix
F	Yield criterion function - a scalar
$\{F\}$	Applied nodal forces (global)
$[G]$	Matrix relating element strains to edge strains
h	Depth below ground surface
H	Depth of water
H'	Hardening function
I	Moment of inertia or second moment of area
$[I]$	The unit or identity matrix
$[J]$	The Jacobian matrix
k	Coefficient of earth pressure at rest
k_x, k_y	Permeability in the x and y directions respectively
$[k]$	Permeability matrix
K	Total no. of structure degrees of freedom
K_t	Stress concentration factor
$[K]$	Structure stiffness matrix (global)
$[K_T]$	Tangential stiffness matrix
$[K_\sigma]$	Initial stress matrix
$[K_o]$	Linear stiffness matrix
$[K_L]$	Initial or large displacement matrix
$[K_N]$	Natural stiffness matrix
L	Length
$\lceil L \rfloor$	A diagonal matrix of reciprocal of side lengths
M	No. of degrees of freedom per element
n	Total no. of equations
N	Total no. of modified elements
NST	Natural stiffness triangular element

[N]	Matrix of shape or interpolation functions
{p}	Concentrated forces or point loads
P	Internal force due to applied loads
{P}	Vector of internal forces due to applied loads
{q}	Surface tractions
Q	Total no. of reaction components
QUAD4	4-node isoparametric quadrilateral element
QUAD8	8-node isoparametric quadrilateral element
r	Denotes r'th iteration, used as a superscript
{r}	Vector of scale factors
R	Variation factor
{R}	Vector of variation factors
$\{R_c\}$	Vector of condensed variation factors
$[R_{cI}]$	Matrix of condensed variation factors for unit loads
t	Transition curve
{T}	Vector of reactions due to applied loads
[T]	Matrix of reactions due to unit loads
u	Unaffected nodes
v	Total no. of modified degrees of freedom
{v}	Velocity vector
W	Total no. of modified degrees of freedom
x_1, x_2	Design variables
{X}	Coordinates of a point in a body

Greek symbols

α	A factor defining changes in area, thickness or elastic modulus
$\lceil \alpha \rfloor$	A diagonal matrix defining material changes
β	A factor defining changes in moment of inertia
γ	Unit weight of soil
Γ	Boundary
δ	Displacement component
$\{\delta\}$	Vector of nodal displacements
$[\delta]$	Matrix of displacements due to unit loads
Δ	Changes in matrices or vectors
$\{\varepsilon\}$	Strain vector

$\{d\varepsilon_e\}$	Incremental elastic strain vector
$\{d\varepsilon_p\}$	Incremental plastic strain vector
θ	Angle
κ	A hardening parameter
λ	Load factor
ν	Poisson's ratio
σ	Axial stress
σ_p	Maximum principal stress
σ_s	Standard stress
σ_y	Yield stress
$\{\sigma\}$	Stress vector
Σ	Summation
ϕ	Potential or head
$\{\psi\}$	Vector of residual or out of balance forces
Ω	Domain

Other symbols

*	When used as a superscript it denotes new or modified quantities. When used as a subscript it denotes virtual quantities.
i,j,k	When used as a subscript it denotes coefficient of a matrix or vector
—	An overbar denotes expansion of a matrix to a larger size

Abbreviations of nonlinear analysis techniques

The Newton-Raphson method and its degenerate forms

1. **IS technique** : The initial stiffness technique by the Newton-Raphson method.

2. **TS technique** : The tangential stiffness technique by the Newton-Raphson method.

3. **TSV technique** : The variant form of the tangential stiffness technique by the Newton-Raphson method.

4. **ITS technique** : The initial/tangential stiffness technique by the Newton-Raphson method. The tangential stiffness is evaluated at the first iteration of each load increment.

5. **ITSV technique** : The variant form of the initial/tangential stiffness technique by the Newton-Raphson method. At the first iteration the elastic stiffness is used, and the tangential stiffness is evaluated at the second iteration.

6. **ITS2 technique** : The initial/tangential stiffness technique by the Newton-Raphson method. The tangential stiffness is evaluated at the second iteration of each load increment.

The proposed techniques by the theorems of geometric variation

1. **ISG technique** : The initial stiffness technique by the theorems of geometric variation.

2. **ISVG technique** : The variant form of the initial stiffness technique by the theorems of geometric variation.

3. **TSG technique** : The tangential stiffness technique by the theorems of geometric variation.

4. **TSVG technique** : The variant form of the tangential stiffness technique by the theorems of geometric variation.

5. **ITSG technique** : The initial/tangential stiffness technique by the theorems of geometric variation. The tangential stiffness is evaluated at the first iteration of each load increment.

6. **ITSVG technique** : The variant form of the initial/tangential stiffness technique by the theorems of geometric variation. The elastic stiffness is used at the first iteration, and the tangential stiffness is evaluated at the second iteration.

7. **ITSG2 technique** : The initial/tangential stiffness technique by the theorems of geometric variation. The tangential stiffness is evaluated at the second iteration of each load increment.

CHAPTER 1

INTRODUCTION

1.1 General remarks

During the past few decades the development of computer based techniques in structural and stress analysis have made a profound impact on the design of structures. One such technique now widely used is the finite element method.

The finite element method is a generalisation of the familiar matrix methods of structural analysis for skeletal structures. An early survey of matrix analysis appears in reference [1]. The modern technique as it is known today was developed independently by Argyris and Kelsey[5] and presented in the classic paper of Turner and his co-workers[141] in the 1950's. The need to develop a better analysis technique arose because of the demands in the aircraft industry for analysing complex structural systems. Obviously their derivations were based on the theory that has been developed earlier in continuum mechanics and matrix methods of analysis. Their significant contributions were presented in matrix forms which may be easily implemented on a computer. This was one of the main inherent strengths and popularity of the method. The term 'finite element' was first used by Clough[31] as late as 1960. Spooner[132] gives an interesting account of these developments as well as the computer programs that were developed to perform these analyses. A historical account of the development of the finite element method may also be found in either the text of Martin and Carey[88], or Huebner and Thornton[60]. Both texts also trace the development of the technique in other areas of applied science and engineering.

The early developments of the finite element method were mainly on linear elastic structural analysis. The main areas of research at this time were: the development of efficient elements; whether compatibility (or equilibrium) across element boundaries were satisfied; the question of convergence; the accuracy of the solution; the errors that can be expected; efficient equation solvers and the range of areas ~~of~~ where the method may be used profitably. At the same time the method was placed on a ~~more~~ firmer mathematical foundation and became recognised as a numerical technique for solving partial differential equations. Its advantages over that of the finite difference technique were also highlighted. The advantages included the ease of analysing problems with complex boundary conditions and irregular geometries. The method also provided a better physical hindsight for engineers to grasp. insight

Indeed it is true to say that today the results obtained from the finite element method are reliable, at least for a linear elastic analysis. Many engineers have a tendency (and also the accessibility of software packages of awesome power) to use the method as a 'blackbox' without sufficient understanding of the concepts, theories and limitations of the method. In linear elastic analysis this is reasonably safe but still requires experience and intuition for example in the type of elements or grading of the finite element mesh to be used effectively in a particular problem.

However in the field of nonlinear analysis, the problem is still deceptively difficult and uncertain as mentioned by Oden and Bathe[111]. In a rather amusing article, Kovach[77] mentioned that in nature all problems are nonlinear. By making simplifying assumptions the nonlinear problem is reduced to a more manageable form. The finite element method ~~have~~ s at last begun to surmount this nonlinear barrier. The two main types of nonlinearity widely recognised in structural mechanics are ~~that~~ those of geometrical and material nonlinearities. The ever decreasing computer costs have meant that nonlinear analysis is becoming more attainable, and many finite element codes for such analysis have been produced[104]. By comparison nonlinear analysis is still many times more expensive than linear analysis and there is no guarantee that a unique solution can always be found if multiple equilibrium states exist. Therefore the economics of nonlinear analysis must be justified before fully embarking on such an analysis. This means that studies to find better ways to solve nonlinear problems is certainly relevant.

Finally one should not become too impressed by the sheer power of 'supercomputers' to solve our problems as expressed by Oden and Bathe[111]. Too much reliance on computer results leads to complacency, over confidence and even arrogance. There are still many uncertainties in analysis, for example in the validity of the material properties used or in the distribution and magnitude of loadings. The answers obtained from a computer will only describe ~~about~~ the behaviour of the mathematical model. One only has to compare with experimental evidence on how real structures behave to see that there is still much more to learn. However the model should not be so crude that the results are practically worthless. Computation should always assist engineering judgement but not replace it.

1.2 Outline of problem

The efficiency of the solution techniques for nonlinear analysis is still an intense area of activity. Gallagher[44], Clough[30] and Zienkiewicz[155,156] have all indicated that there is still much more to be done in terms of producing efficient and reliable solution algorithms. At the same time the present knowledge of material properties beyond the linear behaviour needs to be expanded so that more plausible constitutive relationships may be used in the analysis. Such constitutive relationships should be applicable to a whole range of load conditions as well as being 'path dependent'. Although nonlinear analysis is expensive, it is a useful design aid, for example in predicting the collapse behaviour, where the loading history of the structure may be traced. The present day ultimate design codes rely heavily on empirical formulas obtained from experimental results. Therefore the use of nonlinear finite element analysis may provide a more rational approach which may be used ~~as~~ to a compliment to ultimate design codes. In addition, nonlinear analysis is particular ly important for slender and thin structures, such as shells or plates where information on their behaviour may be limited. If nonlinear analysis is to be used as a design tool a key prerequisite is an efficient computational scheme.

This thesis is concerned with the development of efficient solution algorithms in linear and nonlinear analysis. The solution algorithm to be investigated is a reanalysis technique which is widely used in structural optimization. Structural optimization techniques are widely used in the aircraft industry where the relentless drive for minimum-weight design and maximum safety is a dominating influence. To achieve an 'optimal design' should be the goal of every engineer.

Nonlinear analysis and optimization design are similar because both involve changes in the original structure. These changes may be in the material properties or in the geometry of the original structure. Efficient reanalysis techniques have been particularly well developed in the field of optimization design where repeated analysis of a structure undergoing repeated modification is required. The use of these techniques for nonlinear analyses are not very common and hence it is one area where its potential may be exploited.

1.3 A review of a reanalysis technique

If a structure undergoes changes, then repeated application of the matrix displacement method is required. For large structures (not in the physical sense but rather in the number of unknowns to be solved) this would require a large number of simultaneous equations to be reformed and solved. This is particular true in nonlinear analysis and optimization design, where the structure is progressively modified. In large problems, the solution requires a lot of computational effort even though small changes are introduced. Hence it is worthwhile to develop reanalysis techniques for such problems.

In an attempt to avoid repeated analyses, a number of reanalysis techniques[Appendix I] have been developed. These reanalysis techniques predict the response of the modified structure using the original analysis of the structure in such a way that the computational effort is less than that required for a fresh analysis. There are many reanalysis techniques that are applicable to nonlinear analysis as reviewed in Appendix I. However it is impossible to investigate every one. Hence only the technique based on the theorems of structural and geometric variation developed by Majid et.al.[84,85,86,87] and Topping et.al.[26,139] respectively will be used in this thesis. These theorems give an exact response of the structure after modifications.

1.3.1 Formulation of reanalysis

There are many reanalysis techniques, some of which are based on the force (or flexibility) method, displacement (or stiffness) method or mixed method of analysis. The equations that are derived here are expressed in terms of the displacement method because of its ease of computer implementation.

Consider the analysis of a structure by the displacement method which results in the familiar equilibrium equations:

$$[K]\{\delta\} = \{F\} \tag{1.1}$$

where:

$[K]$ = structure stiffness matrix;

$\{\delta\}$ = vector of nodal displacements; and

$\{F\}$ = vector of nodal applied loads.

Equation (1.1) may be solved for $\{\delta\}$ and hence the strains and stresses of the structure may be calculated. The formation of equation (1.1) in the context of the finite element method will be given in Chapter 2.

If during the design procedure certain constraints are violated (for example the displacements or stresses are larger than the permissible values) or a more economic design is required (such that some of the constraints are just satisfied), then one has to redesign by changing the structure. As a result of the redesign, the change in structure stiffness matrix is $[\Delta K]$ and the corresponding change in the displacement vector is $\{\Delta\delta\}$. The new structure stiffness matrix and corresponding displacements are then given by:

$$[K^*] = [K] + [\Delta K] \tag{1.2}$$

$$\{\delta^*\} = \{\delta\} + \{\Delta\delta\} \tag{1.3}$$

where:

$[K^*]$ = the new structure stiffness matrix; and

$\{\delta^*\}$ = the new nodal displacement vector.

The new overall equilibrium equations are:

$$[K^*]\{\delta^*\} = \{F\} \tag{1.4}$$

assuming that the applied loads do not change in magnitude or direction. For local modifications $[\Delta K]$ will be a highly sparse matrix, that is it contains only a small number of nonzero columns and rows. Substituting equations (1.2) and (1.3) into (1.4) results in:

$$([K] + [\Delta K])(\{\delta\} + \{\Delta\delta\}) = \{F\} \tag{1.5}$$

One possibility is to solve equation (1.4) or (1.5) directly and this may prove to be economic for a large number of elements undergoing modifications. Usually the modifications are small and localised in nature and hence a reanalysis technique may be used effectively.

The aim of reanalysis techniques is to obtain $\{\delta^*\}$ using the known values of $\{\delta\}$ by solving equation (1.1) or those from previous analyses. Once $\{\delta^*\}$ is known the modified strains and stresses may be easily computed. Therefore the solution of equation (1.4) or (1.5) is avoided.

1.3.2 The theorems of structural variation

The theorems were first formulated by Majid and Elliott[84] for truss structures in 1973. Explicit relationships were established to predict the response of the modified structure from the original one, when the area of one or more of its members were varied. These relationships were proposed as the "theorems of structural variation".

The technique may be explained by considering the analysis of a truss structure. If during the analysis, the area of the i'th member is modified by an amount dA_i, then:

$$A'_i = A_i + dA_i \tag{1.6}$$

where:

A'_i = modified area;

A_i = original area; and

dA_i = change in area.

A factor α may be defined as:

$$\alpha = dA_i/A_i = (A_i' - A_i)/A_i \tag{1.7}$$

α is positive if the member increases in size, and negative if it decreases in size. Substituting equation (1.7) into (1.6):

$$A_i' = (1 + \alpha)A_i \tag{1.8}$$

If a member is totally removed, then $A_i'=0$ and $A_i=-dA_i$, therefore $\alpha=-1$.

The force in the i'th member due to the external loadings is denoted by P_i. The forces in the N members of the structure are given by $\{P_1, P_2 .... P_N\}$. The i'th member is now split into two members of area A_i' and $-dA_i$ as shown in figure 1.1(ii).

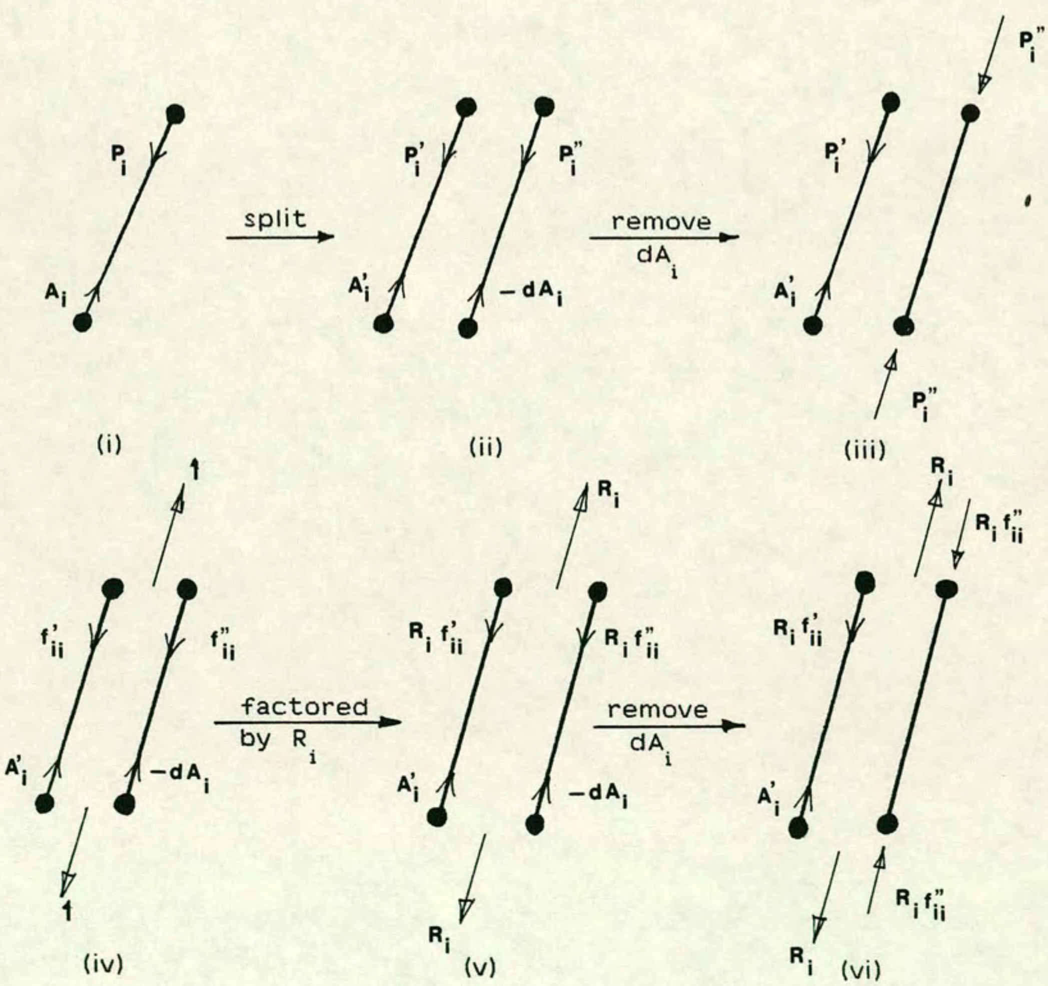


Figure 1.1

The member forces are then P_i' and P_i'' respectively, provided that equilibrium and compatibility are satisfied at the ends of the member. The conditions are:

$$P_i = P'_i + P''_i \tag{1.9a}$$

$$P_i/A_i = P'_i/A'_i = P''_i/(-dA_i) = \sigma \tag{1.9b}$$

where σ = axial stress of i'th member

From equations (1.9b) and (1.7) the split member forces are:

$$P'_i = (1 + \alpha)P_i \tag{1.10a}$$

$$P''_i = -\alpha P_i \tag{1.10b}$$

The forces in the other members of the structure are unchanged if the i'th member is modified to A'_i (by removal of dA_i), provided that an equal and opposite forces, P''_i are applied at the ends of the member as in figure 1.1(iii). However the analysis of the modified structure is required without application of these forces. In other words the net external forces at the ends of the i'th member must be zero.

The P''_i forces are out of balance forces due to the modification. To equilibrate these forces, another analysis is required. The analysis consists of a pair of unit loads applied at the ends of the i'th member, and the force in this member will be denoted by f_{ii}. The forces in the other members are then equal to $\{f_{1i}, f_{2i} \ldots\ldots f_{Ni}\}$. Similarly the forces in split members as shown in figure 1.1(iv) are:

$$f'_{ii} = (1 + \alpha)f_{ii} \tag{1.11a}$$

$$f''_{ii} = -\alpha f_{ii} \tag{1.11b}$$

The unit load analysis is now factored by an amount R_i, and the forces are similarly factored as in figure 1.1(v). Removal of member dA_i requires an equal and opposite force of $R_i f''_{ii}$ as in figure 1.1(vi). By superimposing the loading conditions of figure 1.1(iii) (due to external loadings) and 1.1(vi) (due to factored unit load), the equilibrium condition at the end of the member is given by:

$$R_i - R_i f''_{ii} - P''_i = 0 \quad (1.12)$$

such that member dA_i is removed.

Substitution of equations (1.10b) and (1.11b) gives:

$$R_i = -\alpha P_i/(1 + \alpha f_{ii}) \quad (1.13)$$

The factor R_i is called the variation factor of the i'th member. The force in the modified member A'_i is then:

$$P^*_i = P'_i + R_i f'_{ii} \quad (1.14)$$

and substitution of equations (1.10a),(1.11a) and (1.13) gives:

$$P^*_i = P_i(1 + \alpha)/(1 + \alpha f_{ii}) \quad (1.15)$$

where P^*_i = modified force of i'th member.

The forces in the unaltered m members are:

$$P^*_m = P_m + R_i f_{mi} \quad (1.16)$$

where:

P^*_m = modified force of m'th unaltered member; and

P_m = original force of m'th unaltered member.

Note that m is a subset of N and is equal to N-1 since only one member is varied.

Equations (1.15) and (1.16) are the first theorem of structural variation. The variables in these equations are obtained from the analysis of the original structure subjected to the external loadings and unit load on the i'th member.

The displacements of the modified structure may be similarly obtained by superposition:

$$\delta^*_k = \delta_k + R_i \delta_{ki} \quad (1.17)$$

where:

δ_k^* = modified k'th displacement;

δ_k = original k'th displacement due to external loads;

δ_{ki} = original k'th displacement due to i'th unit load;

and total number of k is equal to the structure's degrees of freedom. Equation (1.17) is the second theorem of structural variation.

If all members of the structure are changed proportionally, then the modified structural stiffness matrix is:

$$[K^*] = (1 + \alpha)[K] \tag{1.18}$$

From equation (1.4):

$$\{\delta^*\} = [K^*]^{-1}\{F\} \tag{1.19}$$

and substitution of (1.18) into (1.19) gives:

$$\{\delta^*\} = \frac{1}{(1+\alpha)}[K]^{-1}\{F\} \tag{1.20}$$

and substitution of (1.1) into (1.20) gives:

$$\{\delta^*\} = \frac{1}{(1+\alpha)}\{\delta\} \tag{1.21}$$

This is the third theorem of structural variation.

If further modification to the member is required, then the results of the unit load analysis must be modified. This may be undertaken by replacing the P's in equations (1.13),(1.15) and (1.16) by f's. The effect of this is to treat the unit load analysis as external loadings. Similarly the δ_k's must be replaced by δ_{ki} in equation (1.17). This procedure allows for the modification of more than one member at a time. It is a sequential approach which requires updating the unit load analyses.

The theorems were further extended by Bakry[14] for the simultaneous modifications of more than one member. If there are n members to be modified by amounts $\alpha_1, \alpha_2 \ldots\ldots \alpha_n$, then n equilibrium equations may be formulated. These equilibrium equations are obtained by considering n pairs of unit load analyses at the ends of the members. By scaling these analyses by n variation factors, $R_1, R_2 .....R_n$, the n equilibrium equations are of the form:

$$
\begin{aligned}
-R_1(1+\alpha_1 f_{11}) - R_2\alpha_1 f_{12} \ldots\ldots -R_n\alpha_1 f_{1n} &= \alpha_1 P_1 \\
-R_1\alpha_2 f_{21} - R_2(1+\alpha_2 f_{22}) \ldots\ldots -R_n\alpha_2 f_{2n} &= \alpha_2 P_2 \\
&\cdot \\
&\cdot \\
&\cdot \\
&\cdot \\
&\cdot \\
-R_1\alpha_n f_{n1} - R_2\alpha_n f_{n2} \ldots\ldots -R_n(1+\alpha_n f_{nn}) &= \alpha_n P_n
\end{aligned}
\qquad (1.22)
$$

Equation (1.22) may be rewritten in compact form as:

$$- R_i - \alpha_i \sum_{j=1}^{n} R_j f_{ij} = \alpha_i P_i \qquad i = 1.....n \qquad (1.23)$$

The simultaneous equations (1.23) are solved for the variation factors R_i.

The forces in the modified members are evaluated as:

$$P_i^* = (1+\alpha_i)P_i + (1+\alpha_i)\sum_{j=1}^{n} R_j f_{ij} \qquad (1.24)$$

and for the unaltered members:

$$P_m^* = P_m + \sum_{j=1}^{n} R_j f_{mj} \qquad (1.25)$$

The number of unaltered m members is equal to N-n.

The modified displacements are:

$$\delta_k^* = \delta_k + \sum_{j=1}^{n} R_j \delta_{kj} \qquad (1.26)$$

The simultaneous modifications allow groups of members to be modified at the same time without modifications to the unit load analyses. This is advantageous and also an efficient technique.

Another important aspect of simultaneous modifications, is that equations (1.23) to (1.26) may be applied to finite element problems. This was presented by Topping[138] for modification of a single element. For example, consider a 4-node quadrilateral element with two degrees of freedom at each node. If the thickness, t of the e'th element is modified, then a factor $\alpha = dt/t$ may be defined. To balance the forces at the nodes due to the modification, eight unit load analyses (for each degree of freedom) are required. By scaling these unit load analyses and using superposition eight equilibrium equations may be formed. These equations are similar to equations (1.23) to (1.26) and are given as:

$$- R_i - \alpha \sum_{j=1}^{8} R_j f_{ij} = \alpha P_i \qquad i = 1......8 \tag{1.27}$$

The modified nodal forces of the e'th element are:

$$P_i^* = (1 + \alpha)P_i + (1 + \alpha)\sum_{j=1}^{8} R_j f_{ij} \tag{1.28}$$

and the nodal forces in the unaltered elements are:

$$P_i^* = P_i + \sum_{j=1}^{8} R_j f_{ij} \tag{1.29}$$

The modified nodal displacements are:

$$\delta_k^* = \delta_k + \sum_{j=1}^{8} R_j \delta_{kj} \tag{1.30}$$

As mentioned in reference [138] for more than one element, the sequential approach must be used. This is time consuming when there are many elements to be varied. In Chapter 3 the extension for modification of more than one element is discussed. Furthermore an efficient technique to form equation (1.27) is derived.

Atrek[12] derived a simplified form of equation (1.24) using matrix notation. The relation between the modified forces are uncoupled from the variation factors. The resulting equation involves less computations and may lead to considerable economy when the number of modified members are large.

The theorems may also be applied to frame structures as presented by Majid, Saka and Celik[86,87]. Here an original member of moment of inertia I, is replaced by a non-prismatic member with moment of inertia I_1' and I_2' at the ends of the member. Then two factors may be defined as $\beta_1 = (I_1' - I)/I$ and $\beta_2 = (I_2' - I)/I$. Two unit load analyses are required with unit moments applied at the ends of the member. By scaling the resulting moments and using superposition, two equilibrium equations are formed. Bakry's[14] technique can then be used to evaluate the two variation factors, the modified bending moments and displacements. If $\beta_1=-1$ and $\beta_2=-1$, the moment of inertia of the member is reduced to zero. It becomes a pin-jointed member and the equations developed earlier for truss structures may then be used. This procedure is used for the elastic-plastic analysis of frame structures as will be mentioned in section 1.3.4. Details of the derivation may be found in references [86,87].

1.3.3 The theorems of geometric variation

The theorems of structural variation are applicable only to changes in the structural properties such as moment of inertia, cross-sectional area or the elastic modulus. Such changes may generate variations in the topology of the structure, but the joint (or node) coordinates are kept constant.

An important step forward for variations of the structural geometry was developed recently by Topping, Majid and Chan[26,139]. These new theorems are hence called the theorems of geometric variation. In references [26] and [139] the theorems of geometric variation were formulated for truss structures. In the original derivation, if a joint is varied then the members connected to that joint will suffer changes in length and orientation. These two changes result in unbalanced forces which must be compensated for if the joint is to be varied. Unit load analyses for each degree of freedom must be scaled and superimposed with the external loads to form the equilibrium equations. These scale factors may be obtained by solving the equilibrium equations, and the modified forces and displacements obtained by superposition. These theorems of geometric variation are also applicable to changes in the structural properties. Therefore the effect of varying the geometry and properties may be investigated at the same time.

More details of these theorems of geometric variation and their extension to finite element problems will be discussed in Chapter 3. The original derivation of these theorems are somewhat confusing and do not lend themselves to an efficient technique. In Chapter 3 some clarifications and simplifications are outlined. The similarities and differences between these two theorems are also discussed.

1.3.4 The use of the theorems of structural variation

The initial use of these theorems were in the field of optimization of truss structures[84,85]. They were used in changing and designing the topology of a structure using mathematical programming techniques. From the case studies, the use of the theorems facilitate the analysis procedure without a fresh analysis everytime. The optimization of frame structures using the theorems were also presented in references [86] and [87].

Majid, Saka and Celik[87,24,25,83] have also applied the technique for the elastic-plastic analysis of rigid steel and concrete structures when calculating the collapse loads. Hinges were inserted into the frame at each point where the internal moment equalled the plastic moment of resistance. The theorems were then used to analyse the resulting modified structure. This process was continued until the collapse load was reached. The nonlinear material behaviour was modelled by representing the stress-strain and moment-curvature diagrams as a series of straight lines. The slope of these lines represented the new material properties as the loading was incremented from one level to the next.

This thesis will also investigate the use of these theorems in nonlinear analysis. However it differs in two ways: firstly the structure involved is assumed to be a finite element idealisation, and secondly the method of solution is a reformulation of the well known and tested Newton-Raphson procedures.

1.3.5 Final comments

In all the references cited so far on the theorems of structural and geometric variation, there was no mentioned of the efficiency based on operation counts or CPU (central processor unit) time required by the technique. The efficiency tests should be based on a comparison with a fresh analysis. Such tests are of utmost importance in selecting these theorems as an efficient reanalysis technique.

Consequently a major effort into the investigation of the efficiency of both theorems are presented in Chapters 5, 6, 7 and 8. In addition potential applications of the technique in linear elastic and nonlinear analysis of finite element problems are also indicated.

1.4 Aims and objectives

The main aims are to investigate the efficiency of these theorems in finite element problems and to use them as solution algorithms in nonlinear analysis. The objective of any solution algorithm that is proposed should possess the desirable properties of :

1. The algorithm must be effective and reliable, and able to handle various types of nonlinear problems. It should be able to analyse for example geometric nonlinearity, material nonlinearity and a combination of both. At the limit points it must be stable and able to deal with the existence of more than one solution.

2. Efficiency is also important especially in terms of the time taken to solve a problem as well as the core storage requirements. This is measured relative to another technique and comparisons will be made with other methods.

3. The ease of implementation into existing computer codes is important from a commercial point of view. Many finite element computer programs have been developed at considerable expense. Therefore the less effort is spent on modifying the existing codes would mean a greater savings since software upgrading and maintenance is expensive.

4. Any solution algorithm should include options to obtain

results to the desired precision, checks on the possibility of divergence and also the use of accelerators to speed the convergence.

There is no solution algorithm that is able to satisfy all the above criteria. Some techniques may be very good for material nonlinear problems and unstable or inefficient for geometrical nonlinear problems. At worse it may even give a false impression that a solution has been obtained. The choice of the optimum method will therefore rests on the user and his experience.

1.5 Layout of thesis

Chapter 1 - This chapter outlines the problem and reviews a reanalysis technique based on the theorems of structural and geometric variation. It also gives the aims and objectives of the thesis.

Chapter 2 - The necessary theory for linear and nonlinear finite element analysis is derived. Only the relevant formulations used in this thesis are given. The linear analysis is used for comparisons with the theorems of structural and geometric variation in Chapters 5 and 6 respectively. The solution algorithms for nonlinear analysis are compared in Chapters 7, 8 and 9.

Chapter 3 - The theorems of structural and geometric variation are derived for linear finite element analysis. These derivations are used in the comparative tests of Chapters 5 and 6. It also forms the basis for nonlinear analysis of Chapter 4.

Chapter 4 - The theorems of structural and geometric variation are derived for nonlinear finite element analysis. The proposed techniques are based on the Newton-Raphson methods. The efficiency of the proposed techniques are investigated in Chapters 7, 8 and 9.

Chapter 5 - Efficiency of the theorems of structural variation in linear analysis is investigated and potential problems illustrated.

Chapter 6 - Efficiency of the theorems of geometric variation in linear analysis is investigated and potential problems illustrated.

Chapter 7 - The theorems of geometric variation are investigated for material nonlinear problems in finite element analysis.

Chapter 8 - The theorems of geometric variation are investigated for geometrical nonlinear problems in finite element analysis.

Chapter 9 - The theorems of geometric variation are investigated for combined material/geometrical nonlinear problems in finite element analysis.

Chapter 10 - General summary, conclusions and future research.

Appendix I - Review of various static reanalysis techniques.

Appendix II - Some equations from the theory of elasticity are used to define the strains, stresses and elasticity matrix. These equations are the necessary ones for the finite element method.

Appendix III - The natural stiffness triangular element used in Chapter 3 is derived.

Appendix IV - A numerical example on the nonlinear analysis of one-dimensional finite elements.

Appendix V - Conversion factors between the foot-pound and S.I. system.

Appendix VI - Computer implementation of the proposed techniques in linear and nonlinear finite element analysis.

CHAPTER 2

THE FINITE ELEMENT METHOD - SOME RELEVANT THEORY

2.1 Introduction

This chapter will briefly describe the use of the finite element displacement method for linear and nonlinear analysis. Detail derivations are not presented here, since many authoritative texts[53,54,60,88,112,154] on finite elements are available for consultation. The main emphasis is on the solution techniques used to solve the finite element equilibrium equations.

The basic steps of the analysis are summarised as follows:

1. The idealisation of the continuum structure into finite elements interconnected by nodes and lines. The type of elements to be used will depend on the displacement field to be specified within each element.

2. The unknowns are the nodal displacements which completely define the response of the structural idealisation. The displacements over each element are then expressed in terms of the nodal displacements by the use of shape or interpolation functions. By invoking the virtual work principle (or other energy principle) the equilibrium equations relating the nodal forces to the nodal displacements for each element may be formulated.

3. The equilibrium equations of the structure are then formed by summing the nodal contributions.

4. By applying the appropriate boundary conditions, the equilibrium equations may be solved for the nodal displacements.

5. The internal strains and stresses are evaluated using the strain-displacement and stress-strain relationships respectively, over each element.

Using each of these steps, a complete solution to the problem is obtained. The procedure is conceptually the same as the stiffness analysis of frame and truss structures. However the difference is in the discretisation and formation of the element stiffness matrix. The element stiffness matrix is approximate because the assumed displacement patterns (by using the shape functions) of the finite element may only approximate the exact displacements of the continuum.

2.2 Linear finite element analysis

In linear elastic analysis deformations are assumed small compared with the dimensions of the **structure** (linear strain-displacement relations) and material behaviour elastic (linear stress-strain relations). The formulations that are given here due to Zienkiewicz[154] form the basis of nonlinear analysis in section 2.3.

Consider the continuum structure of figure 2.1 which is subjected to external loads. The external loads acting onto the body are concentrated forces or point loads {p}, surface tractions {q} and body forces {b}. The domain of the structure is denoted by Ω and the boundary by Γ. Along the boundary, there are some parts which are specified; these are the structural supports.

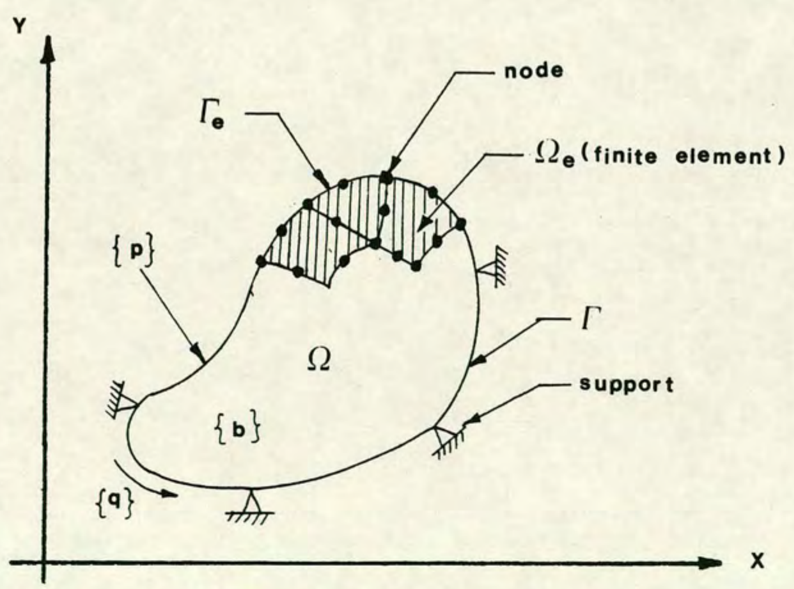


Figure 2.1 Problem domain Ω and boundary Γ

By using the finite element method, the infinite number of degrees of freedom of the continuum is approximated to a finite number. This may be undertaken by first dividing up the continuum into finite elements as shown in figure 2.1. The elements are connected at a finite number of points and a solution is sought at these points. In effect the domain Ω is divided into smaller sub-domains Ω_e and the boundary Γ into sub-boundaries Γ_e. The behaviour of the total domain is the sum of the individual behaviour of the sub-domains. Hence a more complicated problem is reduced to a much simpler one.

2.2.1 Stiffness relations

The action of the loads on the structure of figure 2.1, results in deformation of the structure. The displacements of the structure from the unloaded configuration is denoted by $\{\delta\}$. Similarly the stresses and strains are denoted by $\{\sigma\}$ and $\{\varepsilon\}$ respectively.

Assuming that the applied loads are known; the problem is to solve for the displacements, strains and stresses. The linear theory of elasticity[146] may be used to establish the equilibrium, compatibility, strain-displacement and stress-strain relationships. Subject to the appropriate boundary conditions, an exact analytical solution of these relationships is impossible to obtain, except for the most simple problems. Nevertheless it serves as a useful starting point for the finite element method.

An alternative approach is to use the principle of virtual displacements[154] to express the equilibrium state of the structure. The principle states that the equilibrium of the body requires that for any compatible, small virtual displacements imposed on the body, the total internal virtual work is equal to the external virtual work. This statement is independent of the material behaviour and magnitude of displacements. It is therefore equally valid to linear and nonlinear material or geometrical behaviour. This principle is used to derive the stiffness relations for linear and nonlinear analysis. There are other methods to derive the stiffness relations such as the variational approach or weighted residual methods, but these will not be considered here.

If the body is now subjected to an arbitrary virtual displacement $\{\delta_*\}$ and the strain distribution is $\{\varepsilon_*\}$, then the principle of virtual displacements requires that:

$$\int_\Omega \{\varepsilon_*\}^T \{\sigma\}\, d\Omega = \{\delta_*\}^T \{p\} + \int_\Omega \{\delta_*\}^T \{b\}\, d\Omega + \int_\Gamma \{\delta_*\}^T \{q\}\, d\Gamma \qquad (2.1)$$

The external virtual work is on the right side of equation (2.1) and is equal to the actual applied forces undergoing virtual displacements. The internal virtual work is on the left side and is equal to the actual stresses multiplied by the virtual strains. Equation (2.1) is an expression of equilibrium and contains the compatibility and constitutive relationships. In the finite element method the displacements are considered continuous and compatible, and should satisfy the boundary conditions. The stresses should be evaluated from the displacements using the appropriate constitutive relations. In equation (2.1) the integration is performed over the whole region Ω and boundary Γ. By using the finite element method, the integration instead operates on the sub-domains (or elements) and sub-boundaries. Therefore in equation (2.1) the integration may be written as:

$$\int_\Omega A\, d\Omega = \sum_e \int_{\Omega e} A\, d\Omega_e \qquad (2.2)$$

where A is any function that is integrable in the domain Ω and similarly for the boundary Γ. The summation sign indicates that the total response is the sum of each individual element responses. By using equation (2.2), equation (2.1) is rewritten as the sum of integration of all finite elements:

$$\sum_e \int_{\Omega e} \{\varepsilon_*^e\}^T \{\sigma^e\}\, d\Omega_e =$$
$$\sum_e \{\delta_*^e\}^T \{p^e\} + \sum_e \int_{\Omega e} \{\delta_*^e\}^T \{b^e\}\, d\Omega_e + \sum_e \int_{\Gamma e} \{\delta_*^e\}^T \{q^e\}\, d\Gamma_e \qquad (2.3)$$

The variation of displacements within each element are expressed in terms of nodal values by the shape functions:

$$\{\delta^e\} = [N]\{\delta\} \qquad (2.4)$$

where:

$\{\delta^e\}$ = displacements within the element;

$[N]$ = matrix of shape functions of an element; and

$\{\delta\}$ = nodal displacements of an element.

With the assumption on the displacements in (2.4) the strain distributions within the element is:

$$\{\varepsilon^e\} = [B]\{\delta\} \tag{2.5}$$

where:

$\{\varepsilon^e\}$ = element strains; and

$[B]$ = strain-displacement matrix.

The matrix [B] is composed of derivatives of the shape functions, obtained from the strain-displacement relations[Appendix II]. The stresses within the element are related[Appendix II] to the element strains by:

$$\{\sigma^e\} = [D]\{\varepsilon^e\} \tag{2.6}$$

where:

$\{\sigma^e\}$ = element stresses; and

$[D]$ = elasticity matrix of an element.

For a linear elastic constitutive relationship, [D] is symmetric and only two parameters are needed to completely defined it.

Equations (2.4),(2.5) and (2.6) must also hold for all values of virtual displacements $\{\delta^e_*\}$ and strains $\{\varepsilon^e_*\}$. Substituting these equations into (2.3):

$$\begin{aligned}\{\delta_*\}^T[\textstyle\sum_e\int_{\Omega e}[B]^T[D][B]\,d\Omega_e]\{\delta\} = \{\delta_*\}^T[\ &\textstyle\sum_e[N]^T\{p^e\}\\ &+ \textstyle\sum_e\int_{\Omega e}[N]^T\{b^e\}\,d\Omega_e\\ &+ \textstyle\sum_e\int_{\Gamma e}[N]^T\{q^e\}\,d\Gamma_e\,]\end{aligned} \tag{2.7}$$

In using equation (2.7) the matrices and vectors that are formed must be expanded to the total number of displacement unknowns for matrix formalism during the summation. Hence $\{\delta_*\}$ and $\{\delta\}$ now correspond to the structure's virtual and real displacements. This assemblage process of the element matrices is often referred to as the direct stiffness method.

To obtain the nodal displacements, the virtual displacement theorem is invoked by imposing unit virtual displacements in turn for all displacement components. The equilibrium equations of the assemblage are then:

$$[K]\{\delta\} = \{F\} \tag{2.8}$$

The structure stiffness matrix is given by:

$$[K] = \sum_e \int_{\Omega e} [B]^T [D][B] \, d\Omega_e \tag{2.9a}$$

and the element stiffness matrix $[K^e]$:

$$[K^e] = \int_{\Omega e} [B]^T [D][B] \, d\Omega_e \tag{2.9b}$$

The applied loads $\{F\}$ include the effects of point loads, body forces and surface tractions:

$$\{F\} = \{F_p\} + \{F_b\} + \{F_q\} \tag{2.10}$$

where:

$$\{F_p\} = \sum_e [N]^T \{p^e\} \tag{2.11a}$$

$$\{F_b\} = \sum_e \int_{\Omega e} [N]^T \{b^e\} \, d\Omega_e \tag{2.11b}$$

$$\{F_q\} = \sum_e \int_{\Gamma e} [N]^T \{q^e\} \, d\Gamma_e \tag{2.11c}$$

and equivalent element nodal forces:

$$\{F^e_p\} = [N]^T \{p^e\} \tag{2.12a}$$

$$\{F^e_b\} = \int_{\Omega e} [N]^T \{b^e\} \, d\Omega_e \tag{2.12b}$$

$$\{F^e_q\} = \int_{\Gamma e} [N]^T \{q^e\} \, d\Gamma_e \tag{2.12c}$$

In the computer implementation the element stiffness matrices and nodal force vectors are formed first using equations (2.9b),(2.12a),(2.12b) and (2.12c) respectively in a compact form. The matrices and vectors are the order of the number of element degrees of freedom or displacement unknowns. They may then be assembled using equations (2.9a),(2.10),(2.11a),(2.11b) and (2.11c) to form the structure's equilibrium equations (2.8).

2.2.2 Overall stiffness equations

To describe the behaviour of the structure, the element stiffness and equivalent nodal forces are assembled. The total assemblage is the overall or the structure stiffness matrix which is formed by insisting that compatibility at the element nodes must be satisfied. This means that displacements at common nodes where elements are connected must be the same for all elements. The process may easily be implemented on a computer and it follows directly from matrix structural analysis.

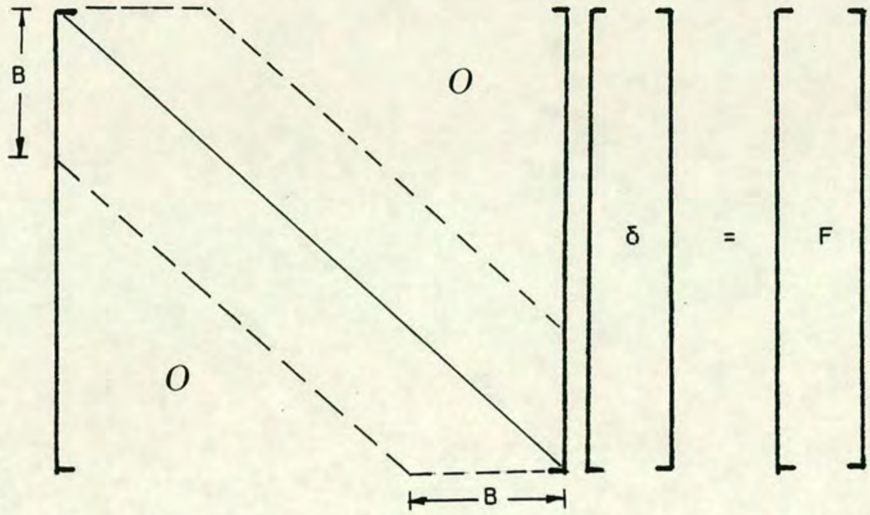
The element stiffness matrix $[K^e]$ relates the nodal forces to the nodal displacements in a compact form. For matrix addition to be valid, $[K^e]$ must be expanded to the structural degrees of freedom, $[\overline{K}^e]$. The expanded element stiffness matrix contains many zero entries except for those directly connected to the element degrees of freedom. The assembly procedure is therefore:

$$[K] = \sum_{e=1}^{M} [\overline{K}^e] = [\overline{K}^1] + [\overline{K}^2] + \ldots\ldots\ldots \tag{2.13}$$

where M = total number of elements in the structure.

The same procedure also applies to the equivalent loads.

In numerical computation this expansion concept is not used at all. Rather a correspondence between the element and structural degrees of freedom is first established, then element matrices are added to their locations in the overall stiffness matrix. The equilibrium equations of the structure are then given by (2.8), and if written in full have the appearance of:



(2.14)

The structure stiffness matrix [K] is sparse and also symmetric about the leading diagonal. The nonzero coefficients are clustered within a bandwidth. Only the semi-bandwidth (including the diagonal), of width B need to be stored. This scheme of storage used in this thesis is discussed briefly in **Appendix VI**.

2.2.3 Solution of stiffness equations

Before equation (2.14) may be solved, the boundary conditions must be applied to render [K] nonsingular. There are a variety of ways of doing this. One method is to rearrange the equations such that the free and restrained degrees of freedom are partitioned. This however destroys the sparsity and banded nature of [K], and is therefore not practical. The method chosen here is that rows and columns corresponding to the restrained degrees of freedom are set to zero and the diagonal to one, and the load vector to the known value of the restrained displacement. The remaining coefficients of {F} are modified by subtracting from it the value of restrained displacement times the appropriate column term of [K]. This procedure is repeated for each restrained displacements and is used with the Gaussian elimination technique described below. It results in the displacements and reactions to be calculated at the same time.

The solution scheme used is the direct tҫechnique of Gaussian elimination. It is a fairly simple technique to implement and also effective. In the process [K] is progressively reduced to a upper triangular form. At the same time {F} is also similarly reduced. Below the leading diagonal of the reduced [K], all coefficients are zero. The solution is obtained by back-substitution where the last equation is solved first and then it proceeds 'upwards'. During the reduction process checks may be made for the positive-definiteness of [K]. This will indicate whether the structure is unstable or the problem is badly modelled.

For multiple load cases, there is no need to form and reduce [K] again. During the first reduction, the Gaussian reduction factors used in making columns below the diagonal zero are stored. Provided that the same structure is used throughout, these reduction factors are used to reduce the subsequent load vectors. The displacements are then obtained by back-substitution. This technique is also useful for nonlinear analysis where the loads are the 'unbalance loads' due to nonlinearities. Another possible technique is to reduce all the load vectors at the same time as [K]. This is more efficient but requires extra storage space. Both techniques are used in this thesis as discussed in **Appendix VI.**

2.2.4 Strain and stress calculations

Once the displacements are known, it is a simple matter to evaluate the strains and stresses using equations (2.5) and (2.6) respectively.

The strains and stresses may be evaluated anywhere within the element. If numerical integration techniques are used to evaluate the element stiffness matrix, the optimal points for evaluating the strains and stresses are the sampling points[154]. This is also convenient because the [B] and [D][B] matrices at each sampling point may be stored during the evaluation of $[K^e]$. To obtain the strains and stresses at the element edges or nodes from the sampling points, extrapolation techniques are used[52].

2.3 Nonlinear finite element analysis

In section 2.2 the finite element formulations were based on linear theory. Displacements of the finite element assemblage were assumed small compared with the dimensions of the structure and that the material behaviour was elastic. Furthermore it was assumed that the boundary conditions do not change during the loading process. These assumptions are valid for a restricted class of problem. The introduction of nonlinearities require a reorganisation of the formulations presented in section 2.2.

Nonlinear problems are characterised by the nonlinear relationship between the applied loads and displacements. In such problems, the principle of superposition of different load cases is no longer valid. The solution techniques are usually of the form of an incremental or iterative procedure which is a sequence of linear analysis. This means repeated linear analysis of the structure is required. It must be pointed out that in linear analysis the solution is unique. This may no longer be the case in nonlinear analysis where the solution obtained might not be the one sought. Nonlinear analysis may require many trial and error computer runs to obtain a solution. Physical understanding and experience of tackling these types of problem is of great help.

In this section only the relevant formulations for nonlinear problems are presented. These will be used in Chapters 7,8 and 9. The important aspect of this section is in the nonlinear solution techniques. Here various forms of the Newton-Raphson procedures are discussed. These procedures are reformulated using the theorems of structural and geometric variation in Chapter 4.

2.3.1 Material nonlinearity

This category of nonlinearity is in the form of a nonlinear stress-strain relationship. The material behaviour may be modelled as nonlinear elastic, hyperelastic or hypoelastic. A comprehensive treatment of these constitutive laws is given by Chen and Saleeb[28].

An important subclass is the elasto-plastic behaviour which will be considered here. Many significant engineering problems fall under this subclass, examples being metals, soils and rocks subjected to high stresses. For purposes of this thesis, strains and displacements are assumed small in elasto-plastic analysis. Accordingly a complete revision of section 2.2 is not required. Hence the stress and strain (or engineering stress and strain) measures used previously are assumed valid. The only change is in the elasticity matrix, [D] which is replaced by the elasto-plastic matrix, $[D_{ep}]$.

To describe the plastic behaviour the incremental or flow theory of plasticity is used[112]. It is assumed that the equations here are for isotropic elasto-plastic solids. In general plastic behaviour may be described by three basic concepts of plasticity theory; the yield condition, the flow rule and the material-hardening rule. Zienkiewicz et.al.[100,159] give a detailed treatment of this problem, and the following is a summary of their work.

2.3.1.1 Yield condition

This yield condition specifies that the onset of plasticity under a state of multiaxial stress or possible combinations of stresses may be expressed as:

$$f(\{\sigma\}) - k(\kappa) = 0 \tag{2.15a}$$

or $$F(\{\sigma\},\kappa) = 0 \tag{2.15b}$$

where:

f,k = some functions of stress and κ respectively;

$\{\sigma\}$ = stress vector; and

κ = a 'hardening' parameter.

F is a scalar function which may be visualised as a surface of n-dimensional space, where n is the number of components of $\{\sigma\}$. There are many yield criteria that have been proposed such as Von Mises, Tresca, Mohr-Columb and Drucker-Prager.

2.3.1.2 Flow rule

The flow rule relates the plastic strain increment to the current stresses and the stress increments subsequent to yielding. According to this rule the strain increments may be derived from a function called the 'plastic potential'. The increments are given by:

$$\{d\varepsilon_p\} = \lambda \frac{\partial Q}{\partial\{\sigma\}} = \lambda \frac{\partial Q(\{\sigma\},\kappa)}{\partial\{\sigma\}} \qquad (2.16)$$

where:

$\{d\varepsilon_p\}$ = incremental plastic strain vector;

λ = proportionality constant to be determined; and

Q = plastic potential function.

Here Q is taken to be identical to F and this is known as associated plasticity. Otherwise the plasticity is non-associated and this leads to unsymmetric elasto-plastic matrices. This requires an equation solver to invert the resulting unsymmetric structure stiffness matrix.

This rule is also known as the normality rule, because the incremental plastic strain vector is normal to the yield surface.

2.3.1.3 Material-hardening rule

This rule modifies the initial yield surface during plastic deformation. There are several rules available and only the simplest is selected which is the isotropic strain hardening rule. The change in the hardening parameter is assumed equal to the amount of plastic work done during plastic deformation:

$$d\kappa = \{\sigma\}^T\{d\varepsilon_p\} \qquad (2.17)$$

This rule may be visualised as a uniform expansion of the yield surface without translation. In some cases strain softening may occur, and the yield surface then contracts. If the yield surface do not change during plastic flow then the behaviour is perfectly plastic.

2.3.1.4 Incremental stress-strain relationship

Equations (2.15),(2.16) and (2.17) are available to form the incremental stress-strain relations that are valid beyond the elastic limit. It is assumed that the total strain increment consists of an elastic and plastic parts:

$$\{d\varepsilon\} = \{d\varepsilon_e\} + \{d\varepsilon_p\} \tag{2.18}$$

where:

$\{d\varepsilon\}$ = total strain increment vector; and

$\{d\varepsilon_e\}$ = incremental elastic strain vector.

By various manipulations of equations (2.6),(2.15),(2.16),(2.17) and (2.18) the elasto-plastic matrix is given by:

$$\{d\sigma\} = [D_{ep}]\{d\varepsilon\} \tag{2.19}$$

where $[D_{ep}]$ = elasto-plastic matrix

and

$$[D_{ep}] = [D] - [D_p] \tag{2.20a}$$

where:

$$[D_p] = [D]\{d\}\{d\}^T[D](A + \{d\}^T[D]\{d\})^{-1} \tag{2.20b}$$

$$\{d\} = \{\partial F/\partial\{\sigma\}\} \tag{2.20c}$$

$$A = H' = \frac{E_T}{1-E_T/E} \tag{2.20d}$$

where:

$[D_p]$ = the plastic part of $[D_{ep}]$;

H' = the hardening function

(slope of stress-plastic strain curve);

E_T = tangent modulus; and

E = elastic modulus.

H' is evaluated from uniaxial tests carried out for the material as shown in figure 2.2. σ_y denotes the yield stress of the material. If $H' = 0$ the material behaviour is perfectly plastic, and there is no strain hardening.

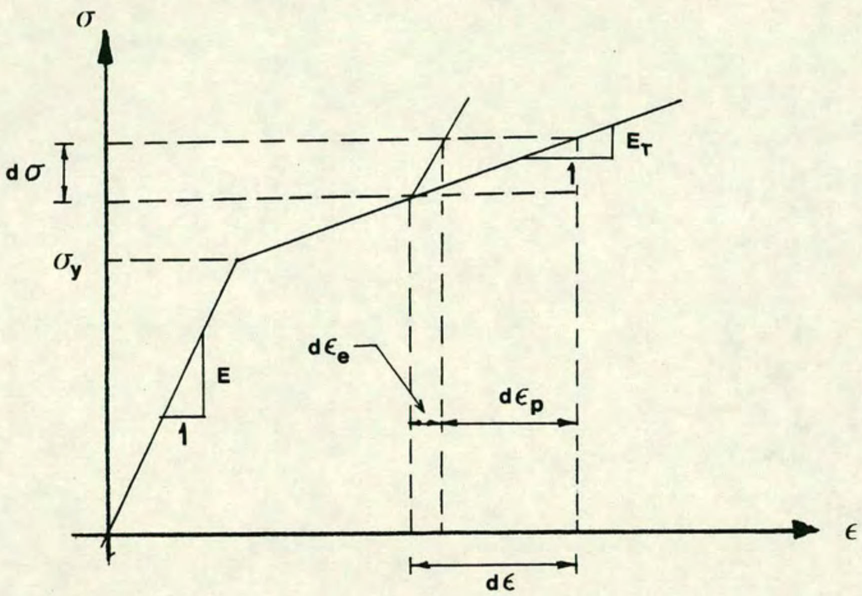


Figure 2.2 Elasto-plastic material behaviour for uniaxial case (showing linear strain hardening)

Equation (2.20a) indicates that plastic action reduces the strength of the material by reducing the magnitudes of the parameters in the elasticity matrix [D].

The solution algorithms to treat plasticity is discussed in section 2.4. For each load increment the tangential stiffness matrix, $[K_T]$ is formed:

$$[K_T] = \int_\Omega [B]^T [D_{ep}][B]\, d\Omega \tag{2.21}$$

Note that in finite element computations the element stiffnesses corresponding to equation (2.21) are formed as in linear analysis.

2.3.2 Geometrical nonlinearity

Geometrical nonlinearities involve a nonlinear form of the strain-displacement relationship. This relationship is given in Appendix II. This may include large strains and displacements which are derived from large changes in the geometry of the deforming body. It is a complex behaviour where different formulations, strain and stress measures are used.

In this chapter only nonlinear problems involving small strains and large displacements are considered. It is also assumed that the stress-strain relationship is linear. The problem is formulated in the Total Lagrangian coordinate system, in which the deformed body is always referred to the original reference coordinate system. Consequently the appropriate strain and stress measures used are the Green's strains and the 2nd. Piola-Kirchoff stresses[Appendix II] respectively. An example of this subclass of nonlinearity is the elastic postbuckling behaviour of structures.

2.3.2.1 Formulation of the equilibrium equations

The essential feature of this analysis is that the equilibrium equations must be written with respect to the deformed geometry which is not known in advance. Here the Total Lagrangian coordinate system is adopted where everything is referred to it.

Let the coordinates of a body in the original undeformed configuration be $\{X\}_o$. Due to the applied loads the body is deformed by an amount $\{\delta\}_n$. The subscript o denotes the original configuration and n the configuration after deformation. The new coordinates of the body is then:

$$\{X\}_n = \{X\}_o + \{\delta\}_n \qquad (2.22)$$

where $\{X\}_n$ = the coordinates after deformation.

By using the principle of virtual work, a similar equation to that of (2.1) may be written as:

$$\int_{\Omega}\{\varepsilon_*\}_n^T\{\sigma\}_n \; d\Omega = \{\delta_*\}_n^T\{p\} + \int_{\Omega}\{\delta_*\}_n^T\{b\} \; d\Omega + \int_{\Gamma}\{\delta_*\}_n^T\{q\} \; d\Gamma \quad (2.23)$$

where:

$\{\varepsilon\}_n$ = Green's strain; and

$\{\sigma\}_n$ = 2nd. Piola-Kirchoff stresses.

In equation (2.23) the subscript n have been omitted from the applied loads. This means that the loads do not change in magnitude or direction during the loading process. This is called conservative loading, otherwise it is non-conservative and an additional stiffness matrix must be added to account for the change.

The equilibrium equations of the structure are then:

$$\{F\} = \int_{\Omega}[B]_n^T[D][B]_n \; d\Omega \; \{\delta\}_n \quad (2.24)$$

Note that the strain-displacement matrix, $[B]_n$ is a nonlinear form because of Green's strain.

Equation (2.24) cannot be solved directly, an incremental procedure using the Newton-Raphson method is used (see section 2.4). Therefore it must be recast in an incremental form. Rewriting it as:

$$\{\psi\}_n = \int_{\Omega}[B]_n^T\{\sigma\}_n \; d\Omega - \{F\} \quad (2.25)$$

where $\{\psi\}_n$ = vector of residual or unbalanced forces.

Thus taking variations of equation (2.25) gives:

$$d\{\psi\}_n = \int_{\Omega}d[B]_n^T\{\sigma\}_n \; d\Omega + \int_{\Omega}[B]_n^T d\{\sigma\}_n \; d\Omega \equiv [K_T]_n d\{\delta\}_n \quad (2.26)$$

The strain-displacement matrix may be divided into linear and nonlinear parts[154]:

$$[B]_n = [B_o]_n + [B_L]_n \quad (2.27)$$

where:

$[B_o]_n$ = linear strain-displacement matrix; and

$[B_L]_n$ = nonlinear strain-displacement matrix.

Taking variations of equations (2.5),(2.6) and (2.27):

$$d\{\varepsilon\}_n = [B]_n d\{\delta\}_n \tag{2.28a}$$

$$d\{\sigma\}_n = [D]d\{\varepsilon\}_n \tag{2.28b}$$

$$d[B]_n = d[B_L]_n \tag{2.28c}$$

Therefore:

$$d\{\sigma\}_n = [D][B]_n d\{\delta\}_n \tag{2.29}$$

Substituting (2.28c) and (2.29) into (2.26):

$$d\{\psi\}_n = \int_\Omega d[B_L]_n^T\{\sigma\}_n \, d\Omega + \int_\Omega [B]_n^T[D][B]_n \, d\Omega \, d\{\delta\}_n \tag{2.30}$$

From the first term on the right side of (2.30) the initial stress matrix is obtained as:

$$[K_\sigma]_n d\{\delta\}_n = \int_\Omega d[B_L]_n^T\{\sigma\}_n \, d\Omega \tag{2.31}$$

where $[K_\sigma]_n$ = initial stress matrix.

From the second term and substitution of (2.27):

$$[K_o]_n + [K_L]_n = \int_\Omega [B]_n^T[D][B]_n \, d\Omega \tag{2.32a}$$

and

$$[K_o]_n = \int_\Omega [B_o]_n^T[D][B_o]_n \, d\Omega \tag{2.32b}$$

$$[K_L]_n = \int_\Omega ([B_o]_n^T[D][B_L]_n + [B_L]_n^T[D][B_L]_n + [B_L]_n^T[D][B_o]_n) \, d\Omega \tag{2.32c}$$

where:

$[K_o]_n$ = the usual linear stiffness matrix; and

$[K_L]_n$ = the initial or large displacement matrix.

The tangential stiffness matrix, $[K_T]_n$ is:

$$[K_T]_n = [K_o]_n + [K_L]_n + [K_\sigma]_n \tag{2.33}$$

The evaluation of equation (2.33) is carried at the element level as in linear analysis.

2.3.3 Combined material/geometrical nonlinearity

A combination of material/geometrical nonlinearity based on the assumptions of sections 2.3.1 and 2.3.2 is possible. In the derivations for the geometric nonlinearity the elasticity matrix [D] is replaced by the elasto-plastic matrix $[D_{ep}]$. The strain and stress measures are the Green's strains and 2nd. Piola-Kirchoff stresses respectively. As pointed out by Zienkiewicz and Nayak[157], this is valid as long as the strains (elastic and plastic) are small.

Consequently the computer implementation requires very little modifications. A unified treatment of large deformation and plasticity have been presented in reference [157] in the context of the finite element method. Such a general formulation will always be correct but it may be argued that the specialised treatment here is more computational efficient and also easier to understand.

2.4 Nonlinear solution techniques

In nonlinear analysis the overall equilibrium equations may be written generally as:

$$[K(\delta)]\{\delta\} = \{F\} \tag{2.34}$$

Here the stiffness matrix is a function of the unknown displacements. In linear analysis the overall stiffness matrix consists of constant coefficients. It may be solved directly to give a unique solution for $\{\delta\}$ as outlined in section 2.2.3. However in this case the direct solution of (2.34) becomes impossible and an iterative process must be adopted. There are numerous schemes for the solution of nonlinear problems[135,136] including; iterative algorithms, incremental methods, self-correcting procedures and minimisation techniques. The method of Newton-Raphson and its degenerate forms will be discussed at some length here. This method is used for comparison with the theorems of structural and geometric variation in Chapters 4,7,8 and 9.

2.4.1 Newton-Raphson methods

The problem of solving (2.34) may be approached from a mathematical point of view[60,110]. Rewriting equation (2.34) as:

$$\{\psi\} = [K(\delta)]\{\delta\} - \{F\} \tag{2.35}$$

For each equation of (2.35):

$$\psi_i(\delta_1,\delta_2.....\delta_n) = \sum_{j=1}^{n} k_{ij}(\delta_1,\delta_2.....\delta_n)\delta_j - F_i \qquad i=1,2.....n \tag{2.36}$$

where:

ψ_i = i'th equation of residual load;

k_{ij} = coefficients of the stiffness matrix;

n = number of equations (structure degrees of freedom);

and ψ_i and k_{ij} are functions of the displacements.

For an exact solution of $\{\delta\}$ it is required that $\{\psi\}=0$. However the exact solution cannot be directly evaluated, but an approximate solution may be obtained. The quality of the approximate solution will be governed by $\{\psi\}$. If it is zero or within some specified tolerance then a solution has been achieved. However multiple solutions may exist in nonlinear problems and hence the solution obtained is largely dependent upon the first approximation.

To begin with, an improved solution is obtained by the use of the Taylor series expansion for n variables:

$$\psi_i(\delta_1+\Delta\delta_1,\delta_2+\Delta\delta_2.....\delta_n+\Delta\delta_n) = \psi_i(\delta_1,\delta_2.....\delta_n) + \sum_{j=1}^{n} \frac{\partial\psi_i}{\partial\delta_j}(\delta_1,\delta_2.....\delta_n)\Delta\delta_j + ... \tag{2.37}$$

neglecting the higher-order terms.

For a solution of (2.37) the left side must vanished, hence:

$$\sum_{j=1}^{n} \frac{\partial\psi_i}{\partial\delta_j}(\delta_1,\delta_2.....\delta_n)\Delta\delta_j = -\psi_i(\delta_1,\delta_2.....\delta_n) \tag{2.38}$$

Equation (2.38) is a set of algebraic equations to be solved for $\Delta\delta_j$'s. In matrix notation this is:

$$[J^r]\{\Delta\delta^r\} = -\{\psi^r\} \tag{2.39a}$$

$$\{\delta^r\} = \{\delta^{r-1}\} + \{\Delta\delta^r\} \tag{2.39b}$$

where:

r = denotes the r'th iteration;

$[J^r]$ = is the Jacobian matrix;

and the coefficients of $[J^r]$ are:

$$J^r_{ij} = \frac{\partial\psi_i}{\partial\delta_j}(\delta^r_1, \delta^r_2 \ldots\ldots \delta^r_n) \tag{2.39c}$$

Mathematically the Jacobian matrix may be visualised as the slope of a n-dimensional surface.

2.4.1.1 The tangential stiffness technique

From the structural analysis viewpoint the Jacobian matrix is the tangential stiffness matrix, $[K_T]$. It is the slope of the load-displacement curve as shown in figure 2.3 for a one degree of freedom problem.

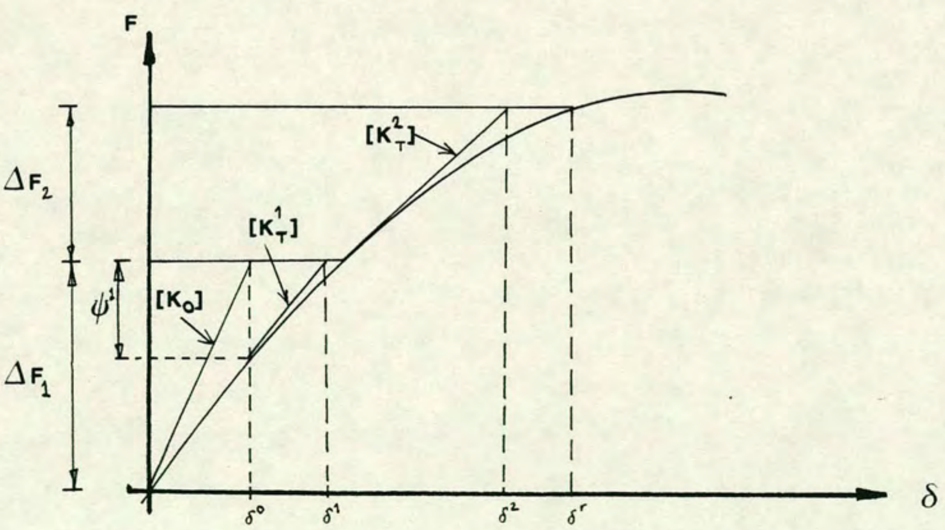


Figure 2.3 Tangential stiffness technique

The formulation of the tangential stiffness matrix for material and geometrical nonlinear problems has been discussed earlier in section 2.3. Therefore equations (2.39) are replaced by:

$$[K^r_T]\{\Delta\delta^r\} = -\{\psi^r\} \tag{2.40a}$$

$$\{\delta^r\} = \{\delta^{r-1}\} + \{\Delta\delta^r\} \tag{2.40b}$$

To start the process an initial estimate of $\{\delta^0\}$ is required; a good estimate is usually the linear solution using the linear stiffness matrix $[K_0]$. The strains and stresses may then be evaluated for the nonlinear structure. The residual loads are given by:

$$\{\psi^r\} = \int_\Omega [B^{r-1}]^T \{\sigma^{r-1}\}\, d\Omega - \{F\} \tag{2.40c}$$

Depending on the sources of nonlinearities either $[B]$,$\{\sigma\}$ or both terms will be recalculated after each iteration.

It should be noted that for each iteration, $[K_T]$ must be formed and reduced to obtain a new solution. Convergence will be achieved when the displacements between successive iterations become tolerably small. Another measure of convergence is that $\{\psi\}$ approaches zero. Since the displacements and residual loads are vectors a global convergence parameter using some vector norms are used. Usually the loads are applied in increments, $\{\Delta F\}$ as shown in figure 2.3.

2.4.1.2 The initial stiffness technique

The recursion equations for this technique are given by:

$$[K_0]\{\Delta\delta^r\} = -\{\psi^r\} \tag{2.41a}$$

$$\{\delta^r\} = \{\delta^{r-1}\} + \{\Delta\delta^r\} \tag{2.41b}$$

$$\{\psi^r\} = \int_\Omega [B^{r-1}]^T \{\sigma^{r-1}\}\, d\Omega - \{F\} \tag{2.41c}$$

In this technique $[K_0]$ is assembled and reduced once. The Gaussian reduction factors are then stored to be used for the reduction of the right hand side of equation (2.41a).

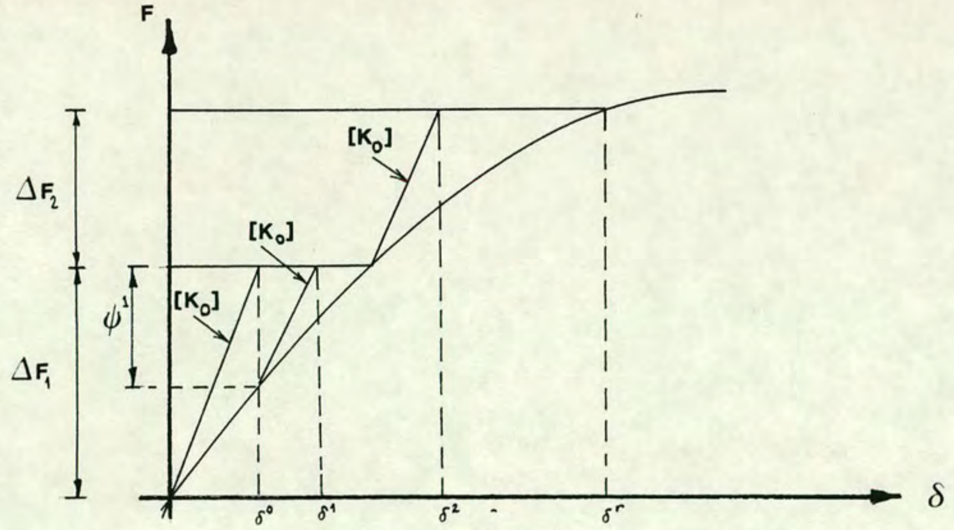


Figure 2.4 Initial stiffness technique

Since the stiffness matrix is only reduced once, each iteration takes very little computational effort. However the technique requires many iterations compared to the tangential stiffness technique. It converges very slowly for highly nonlinear problems and a number of acceleration methods have been devised to accelerate the technique[101]. Figure 2.4 graphically illustrates the technique.

2.4.1.3 The initial/tangential stiffness technique

This is a mixed technique where the advantages of the initial and tangential stiffness techniques are combined. Here the stiffness matrix is changed at selected intervals and kept constant during the load increment as shown in figure 2.5.

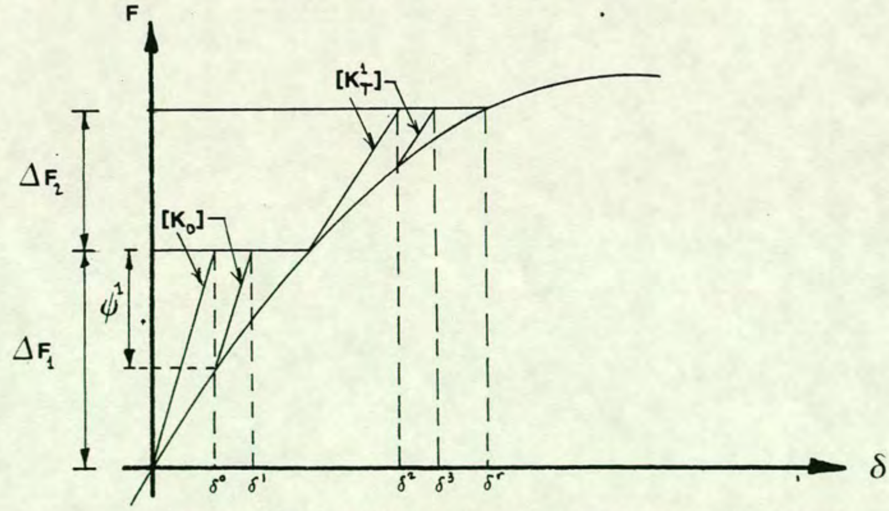


Figure 2.5 Initial/tangential stiffness technique

In figure 2.5 the technique is illustrated for the case when $[K_T]$ is changed at the first iteration of the load increment. It is also possible to consider changing $[K_T]$ at the second iteration.

2.4.1.4 The incremental technique

This was one of the earliest technique used in nonlinear analysis[88]. It was developed purely from physical reasoning of nonlinear problems. Essentially it is the same as the initial/tangential stiffness technique, where $[K_T]$ is changed at the first iteration within a load increment. However no iterations are carried out within the load increment as shown in figure 2.6.

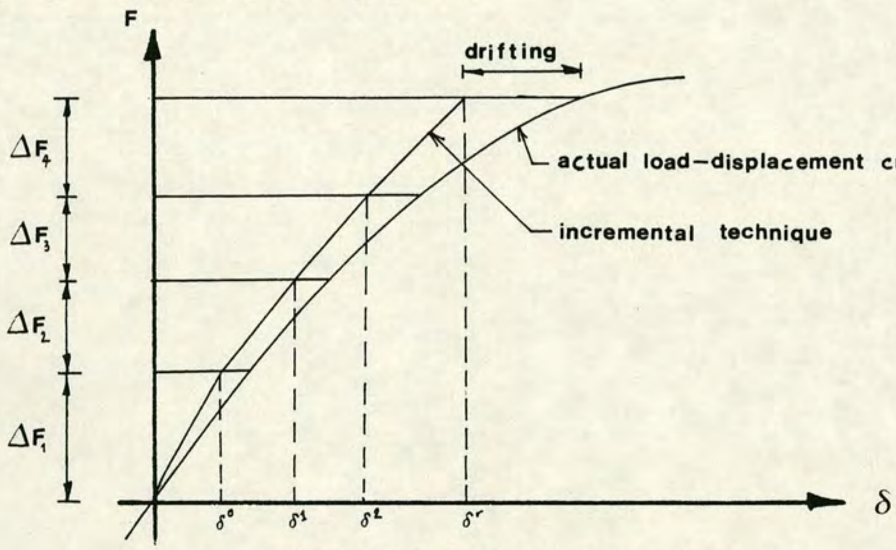


Figure 2.6 Incremental technique

The use of this technique results in 'drifting' of the load-displacement curve. This is because equilibrium is not satisfied at the end of each load increment. Therefore to use this technique effectively the total applied loads must be divided into very small increments. Hence it is generally an expensive technique to use and will not be considered in this thesis.

2.5 Problem and element types used in this research

The type of problems considered here are two-dimensional continuum structures which require only two independent coordinates to specify its geometry and displacement components. These problems may be classified into plane stress, plane strain and axisymmetric solids. The displacement, strain and stress components, and the elasticity matrix may all be found in the texts of [53,54,60,88,112,154].

The displacement components for plane stress, plane strain and axisymmetric solids do not contain the displacement gradients. Consequently the problem is defined as a C(0) problem[154]. In C(0) problems the displacements are continuous across element interfaces. There is no need to satisfy continuity of the gradients or first derivatives of the displacements. The type of elements used are the 3-node triangular element (or constant strain triangular element), the 4-node and 8-node isoparametric elements. The 8-node isoparametric elements are extremely versatile, good performers and are well tried and tested. The derivation of the isoparametric elements and their computer implementation may be found in the texts of Hinton and Owen[53,54,112].

2.6 Some practical considerations

The main effort in using finite element techniques is in generating the input data and interpretation of the output. To obtain reliable results finite element meshes may have many hundreds of nodes and elements. If data is to be prepared by hand, the job would be very tedious, time consuming and prone to errors. Results of displacements, strains and stresses on pages of computer output is also meaningless because it is difficult to visualise the behaviour of the structure.

For these reasons alone, a simple pre- and post-processor computer programs as outlined in **Appendix VI** were developed. The pre-processor is a mesh generation program[53] for generation of 3-node triangular elements, 4- and 8-node quadrilateral isoparametric elements. The graphics package available at ERCC (Edinburgh Regional Computing Centre) was used to draw the mesh on the graphics terminal. A hardcopy may be obtained if the mesh is satisfactory. The used of the Gaussian solver described earlier requires the bandwidth to be small for efficiency. A renumbering program[32] was used to renumber the finite element mesh generated so that the bandwidth was reduced.

Similarly plots of deformed structure and stress contours were obtained using the graphics package.

CHAPTER 3

THE THEOREMS OF STRUCTURAL AND GEOMETRIC VARIATION FOR LINEAR FINITE ELEMENT ANALYSIS

3.1 Introduction

In this chapter the theorems of structural and geometric variation for finite element problems in linear analysis are presented.

In linear analysis the theorems of structural variation have previously been presented for variations in structural properties of one element[138]. In this chapter they are extended for the simultaneous modifications of more than one element. This is important if the theorems are to be used as an efficient reanalysis technique.

The theorems of geometric variation are developed for variations in the coordinates of the nodes of the elements. The formulation follows directly from that presented for truss structures[26,139]. However some clarifications and simplifications of the theorems of geometric variation will reveal the similarities and differences with the theorems of structural variation.

The theorems of structural and geometric variation are both presented in matrix form by considering a simple example. This approach enables an efficient computer implementation of the technique to be identified. The efficiency tests and comparisons with a fresh analysis are discussed in Chapters 5 and 6.

3.2 The theorems of structural variation

The theorems of structural variation for finite element analysis have already been briefly described in Chapter 1. This formulation was only applicable to the modification of one element at a time. If there is more than one element to be modified, the sequential approach must be used. The sequential approach is inefficient because it requires the modification of the unit load analyses of the elements to be modified. This is accomplished by treating the unit loads as applied loads and the approach is therefore time consuming. Equations (1.27) to (1.30) may be reformulated more generally to take into account simultaneous changes in more than one element. This procedure of simultaneous modifications is particularly important for nonlinear analysis where it is highly likely that more than one element may vary.

3.2.1 Simultaneous modifications

The first step is to generalise equations (1.27) to (1.30) that accounted for the structural variation of one element in a finite element idealisation. If N elements are to be modified, the number of equilibrium equations that may be formulated is equal to the total number of degrees of freedom of the N elements. The number of degrees of freedom per element is dependent on both the number of nodes per element and the number of degrees of freedom at each node. Thus defining the change of an element as:

$$\alpha^{e} = \frac{E'_{e} - E_{e}}{E_{e}} \qquad e = 1,2.....N \tag{3.1}$$

where:

E'_{e} = new elastic modulus of e'th element;

E_{e} = original elastic modulus of e'th element; and

N = total number of modified elements.

Note that here the variation of elastic modulus will be considered, but it is equally applicable to other structural properties such as the thickness of the element.

For each degree of freedom in the e'th modified element, the change may be defined as:

$$\alpha_{m} = \alpha^{e} \qquad m = 1,2.....M \tag{3.2}$$

where M = total number of degrees of freedom of the e'th element.

If for example a constant strain triangular element is used in the analysis, then M is six. In equation (3.2) the α's are identified with the degrees of freedom of the element in contrast to equation (1.27) where it was identified with an element.

The equilibrium equations for the nodes of the N modified elements are given by:

$$- R_i - \alpha_i \sum_{j=1}^{v} R_j f_{ij} = \alpha_i P_i \qquad i = 1,2.....v \tag{3.3}$$

where:

R_i = variation factor of the i'th degree of freedom;

f_{ij} = internal force at i'th degree of freedom due to unit load acting at j'th degree of freedom of the modified element;

P_i = internal force at i'th degree of freedom due to applied loads; and

v = total number of degrees of freedom of N elements = (N x M).

Solving for the variation factors $\{R_1, R_2, ...R_v\}$, the new nodal forces of the e'th modified element are given by:

$$P_m^* = (1+\alpha_m)P_m + (1+\alpha_m)\sum_{j=1}^{v} R_j f_{mj} \tag{3.4}$$

and the nodal forces of the unaltered elements are:

$$P_m^* = P_m + \sum_{j=1}^{v} R_j f_{mj} \tag{3.5}$$

Equations (3.4) and (3.5) are the first theorem of structural variation, and the second theorem is given by the modified displacements which are:

$$\delta_k^* = \delta_k + \sum_{j=1}^{v} R_j \delta_{kj} \qquad k = 1,2.....K \tag{3.6}$$

where:

δ_k^* = modified displacement at k'th degree of freedom;

δ_k = original displacement at k'th degree of freedom;

δ_{kj} = displacement at k'th degree of freedom due unit load at j'th degree of freedom of modified element; and

K = total number of degrees of freedom of structure.

The use of equations (3.3) to (3.6) in finite element analysis is very inefficient. First it should be noted that equations (3.4) and (3.5) (or the first theorem of structural variation) are not required. In other words the modified internal forces are of no interest, rather it is the displacements, strains and stresses that are required. The equation involving modified internal forces may therefore be discarded. This is in contrast to truss and frame structures where the internal forces are required[82,84,87]. Secondly the R_i's may be summed into condensed variation factors, R_{ci}'s in equation (3.3). This means that the simultaneous equations (3.3) are reduced and therefore the solution takes less time. This may be illustrated by considering a simple example.

3.2.1.1 Initial analysis

The finite element structure of figure 3.1 has been idealised into constant strain triangular elements. It is a two-dimensional problem with two degrees of freedom at each node.

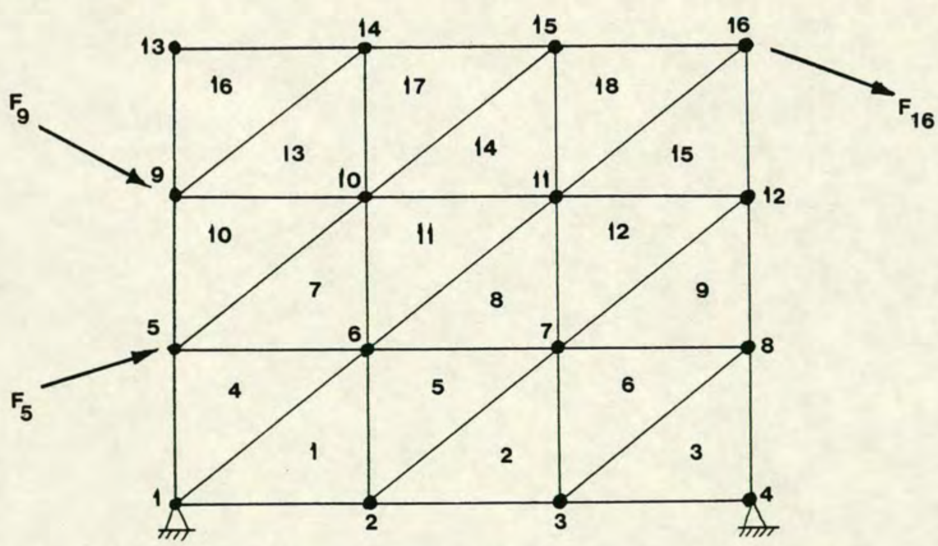


Figure 3.1 Two-dimensional finite element structure

The response of the modified structure is required when the elastic modulus of element 1 is changed from E to E'. To use the theorems, the original structure of figure 3.1 is analysed for unit and applied loads. The unit loads are applied to each node for each degree of freedom of the modified element. For this modified structure the total number of unit loads is 6, which is the total number of degrees of freedom of the element. This analysis is given as:

$$[K][\delta_I \mid \delta] = [I \mid F] \tag{3.7}$$

where:

$[K]$ = structure stiffness matrix;

$[\delta_I]$ = displacements due to unit loads;

$\{\delta\}$ = displacements due to applied loads;

$[I]$ = unit loads; and

$\{F\}$ = applied loads.

Equation (3.7) is for multiple loading conditions. Each column of $[I \mid F]$ corresponds to a loading condition. The columns of the submatrix $[I]$ will contain zeros except those corresponding to a degree of freedom of the modified element which will be one. The formation of $[K]$ in the context of the finite element method was presented in Chapter 2.

The solution of equation (3.7) by the Gaussian elimination technique not only gives the displacements $[\delta_I \mid \delta]$, but also in the reactions at the boundary nodes. These reactions are denoted by:

$$[T_I \mid T] \tag{3.8}$$

where:

$[T_I]$ = reactions due to unit loads; and

$\{T\}$ = reactions due to applied loads.

In the example of figure 3.1, $[I]$ and $[\delta_I]$ are (32 by 6) matrices. The rows of $[I]$ and $[\delta_I]$ correspond to the structure's degrees of freedom and the 6 columns correspond to the 6 unit load analyses. On the otherhand $[T_I]$ is a (4 by 6) matrix. The 4 rows are due to the 4 reaction components at nodes 1 and 4, while the 6 columns are due to the 6 unit load analyses. It was suggested that the nodal forces P_m^* and P_m in equations (3.4) and (3.5) are not required, hence the evaluation of the internal forces from the element stiffnesses and displacements are avoided. For a large finite element structure this saves a considerable amount of computer time. The only internal forces that are required are those for the modified elements to form equation (3.3). A further savings is possible in the calculation of the modified reactions since they may be obtained from a similar equation to (3.6), without evaluation of the internal forces.

If elements 1 and 12, in the structure shown in figure 3.1, are to be modified then the number of unit load analyses required is 12 (which is the number of modified elements multiplied by the number of degrees of freedom per element). However in nonlinear analysis or optimization design, modified elements are usually adjacent to one another. Therefore it is logical to analyse the structure when elements 1 and 4 are modified instead. At first, it would seem that 12 unit load analyses are required at nodes 1,2 & 6 of element 1 and at nodes 1,6 & 5 of element 4. But nodes 1 and 6 are common to both elements and unit load analyses are performed only for nodes 1,2,6 and 5, giving a total of 8 unit loads.

Equation (3.7) is now used to analyse the original structure. The columns of [I] will now depend on the number of modified elements and the nodes connecting them together. Defining the number of unit loads as W then:

$$W = (\text{No. of nodes connecting the modified elements}) \times (\text{No. of degrees of freedom per node}) \qquad (3.9)$$

If the elements are not connected at all, the number of unit loads are (N x M) as given in equation (3.3). Note that W is a subset of the structure's degrees of freedom, K. In other words, the modified parts of the structure are considered as a substructure.

The modified elements that are adjacent to one another results in a reduction of the unit loads to be analysed. This is advantageous in large finite element structures where the solution of equation (3.7) takes most of the time. This procedure however requires extra 'book-keeping' to keep track of the unit loads, but this is not computationally very expensive. When equation (3.7) is solved the reactions of equation (3.8) are also obtained.

3.2.1.2 Analysis of modified structure

After the initial analysis has been performed, the analysis of the modified structure may now proceed. The formulations given here are equally applicable to other elements.

Equations (3.1) and (3.2) are used to define the α's. Equation (3.3) is then used and the f_{ij}'s and P_i's are evaluated using the element stiffness and nodal displacements obtained from $[\delta_I \mid \delta]$. Equation (3.3) is then solved for the variation factors $\{R\}=\{R_1, R_2 \ldots\ldots R_v\}$. It should however be noted that $\{R\}$ may be condensed because some nodes are common to the modified elements. At a common node each R_i's may simply be added to those of the contributions from the other modified elements, to form the condensed variation factors, $\{R_c\}$. This analogy is similar to that of forming the overall stiffness matrix from the element stiffnesses. Therefore equation (3.3) becomes:

$$- R_{ci} - \alpha_i \sum_{j=1}^{W} R_{cj} f_{ij} = \alpha_i P_i \qquad i = 1,2\ldots\ldots W \qquad (3.10)$$

In equation (3.10) the α's, f_{ij}'s and P_i's are also contributions from each modified element. The displacements of the modified structure are then given by:

$$\delta_k^* = \delta_k + \sum_{j=1}^{W} R_{cj} \delta_{kj} \qquad (3.11)$$

The δ_{kj}'s and δ_k's are obtained from $[\delta_I \mid \delta]$.

The reactions of the modified structure may be calculated using a similar equation thus:

$$T_q^* = T_q + \sum_{j=1}^{W} R_{cj} T_{qj} \qquad q = 1,2\ldots\ldots Q \qquad (3.12)$$

where:

T_q^* = modified reaction at q'th degree of freedom;

T_q = original reaction at q'th degree of freedom;

T_{qj} = reaction at q'th degree of freedom due to unit load at j'th degree of freedom of the modified element; and

Q = total number of reaction components.

The T_{qj}'s and T_q's are obtained from $[T_I \mid T]$.

For the complete solution of the finite element problem, the calculation of the strains and stresses are required. The modified strains are evaluated from the strain-displacement relationships. For each element of the structure:

$$\{\varepsilon^*\} = [B]\{\delta^*\} \tag{3.13}$$

where:

$\{\varepsilon^*\}$ = modified element strains; and

$\{\delta^*\}$ = modified element nodal displacements from equation (3.11).

The stress-strain relationships for the original elements are:

$$\{\sigma\} = [D]\{\varepsilon\} \tag{3.14}$$

For the modified structure the new stresses are required. For the unaltered elements of the modified structure and using equation (3.13) the stresses are given by:

$$\{\sigma^*\} = [D][B]\{\delta^*\} \tag{3.15a}$$

and for the modified elements:

$$\{\sigma^*\} = (1+\alpha^e)[D][B]\{\delta^*\} \tag{3.15b}$$

where $\{\sigma^*\}$ = modified element stresses.

In equation (3.15b) a scalar factor of $(1+\alpha^e)$ is required, since the modified elements have a new elastic modulus E'.

The additional equations (3.12),(3.13) and (3.15) complete the analysis of the modified finite element structure. The [B] and [D][B] matrices are usually available as an intermediate step during the evaluation of the element stiffness matrix, $[K^e]$. These matrices are stored for later use with equations (3.13) and (3.15). The procedure that has been outlined is computationally advantageous in terms of execution time and storage requirements compared to the calculation of internal forces.

3.2.1.3 An example

To illustrate the use of these equations, the structure shown in figure 3.1 with elements 1 and 4 modified will be studied. The number of unit load analyses are 8, applied to nodes 1,2,6 and 5 for each degree of freedom. Equation (3.7) is now used to calculate the displacements and reactions. Using equations (3.1) and (3.2):

$$\text{For element 1} \quad \alpha^1 = \alpha_1 = \alpha_2 \ldots\ldots = \alpha_6 \tag{3.16a}$$

$$\text{For element 4} \quad \alpha^4 = \alpha_7 = \alpha_8 \ldots\ldots = \alpha_{12} \tag{3.16b}$$

Before equations (3.3) or (3.10) may be formed, the f_{ij}'s and P_i's must be evaluated from the element stiffnesses and nodal displacements. Equation (3.3) is as shown on the following page in matrix notation (equation (3.17)). The diagonal terms are of the form $(1+\alpha_i f_{ii})$ and the off-diagonal terms $\alpha_i f_{ij}$. Commas have been inserted in the subscripts of the f_{ij}'s in equation (3.17) to distinguish for example $f_{1,11}$ from $f_{11,1}$. Columns 1 and 7 have the same f_{ij}'s and similarly for columns 2 and 8, because node 1 is common. Columns 5 and 9 with 6 and 10 have the same f_{ij}'s because node 6 is common.

The variation factors $\{R_1, R_2, \ldots\ldots R_{12}\}$ are solved and then condensed to $\{R_{c1}, R_{c2}, \ldots\ldots R_{c8}\}$ where:

$$\begin{aligned}
&\left.\begin{aligned} R_{c1} &= R_1 + R_7 \\ R_{c2} &= R_2 + R_8 \end{aligned}\right\} \quad \text{common node 1} \\
&R_{c3} = R_3 \\
&R_{c4} = R_4 \\
&\left.\begin{aligned} R_{c5} &= R_5 + R_9 \\ R_{c6} &= R_6 + R_{10} \end{aligned}\right\} \quad \text{common node 6} \\
&R_{c7} = R_{11} \\
&R_{c8} = R_{12}
\end{aligned} \tag{3.18}$$

$$
\begin{array}{ll}
\text{Element 1} & \begin{array}{l} \text{node 1} \\ \text{node 2} \\ \\ \text{node 6} \\ \\ \end{array} \\
\hline
\text{Element 4} & \begin{array}{l} \text{node 1} \\ \text{node 6} \\ \\ \text{node 5} \\ \\ \end{array}
\end{array}
\left[\begin{array}{cccccc:cccccc}
1+\alpha_1 f_{1,1} & \alpha_1 f_{1,2} & \cdots\cdots & \alpha_1 f_{1,6} & \alpha_1 f_{1,7} & \alpha_1 f_{1,8} & \cdots\cdots & \alpha_1 f_{1,12} \\
\alpha_2 f_{2,1} & 1+\alpha_2 f_{2,2} & \cdots\cdots & \alpha_2 f_{2,6} & \alpha_2 f_{2,7} & \alpha_2 f_{2,8} & \cdots\cdots & \alpha_2 f_{2,12} \\
\vdots & \vdots & & \vdots & \vdots & \vdots & & \vdots \\
\alpha_5 f_{5,1} & \alpha_5 f_{5,2} & \cdots\cdots & \alpha_5 f_{5,6} & \alpha_5 f_{5,7} & \alpha_5 f_{5,8} & \cdots\cdots & \alpha_5 f_{5,12} \\
\alpha_6 f_{6,1} & \alpha_6 f_{6,2} & \cdots\cdots & 1+\alpha_6 f_{6,6} & \alpha_6 f_{6,7} & \alpha_6 f_{6,8} & \cdots\cdots & \alpha_6 f_{6,12} \\
\hdashline
\alpha_7 f_{7,1} & \alpha_7 f_{7,2} & \cdots\cdots & \alpha_7 f_{7,6} & 1+\alpha_7 f_{7,7} & \alpha_7 f_{7,8} & \cdots\cdots & \alpha_7 f_{7,12} \\
\alpha_8 f_{8,1} & \alpha_8 f_{8,2} & \cdots\cdots & \alpha_8 f_{8,6} & \alpha_8 f_{8,7} & 1+\alpha_8 f_{8,8} & \cdots\cdots & \alpha_8 f_{8,12} \\
\vdots & \vdots & & \vdots & \vdots & \vdots & & \vdots \\
\alpha_{12} f_{12,1} & \alpha_{12} f_{12,2} & \cdots\cdots & \alpha_{12} f_{12,6} & \alpha_{12} f_{12,7} & \alpha_{12} f_{12,8} & \cdots\cdots & 1+\alpha_{12} f_{12,12}
\end{array}\right]
\left[\begin{array}{c} R_1 \\ R_2 \\ \vdots \\ R_5 \\ R_6 \\ \hdashline R_7 \\ R_8 \\ \vdots \\ R_{12} \end{array}\right]
+
\left[\begin{array}{c} \alpha_1 P_1 \\ \alpha_2 P_2 \\ \vdots \\ \alpha_5 P_5 \\ \alpha_6 P_6 \\ \hdashline \alpha_7 P_7 \\ \alpha_8 P_8 \\ \vdots \\ \alpha_{12} P_{12} \end{array}\right]
=
\left[\begin{array}{c} 0 \\ 0 \\ \vdots \\ 0 \\ 0 \\ \hdashline 0 \\ 0 \\ \vdots \\ 0 \end{array}\right]
$$

Column	Label
1st	Due to unit load at node 1-x
2nd	Due to unit load at node 1-y
6th	Due to unit load at node 6-y
7th	Due to unit load at node 1-x
8th	Due to unit load at node 1-y
12th	Due to unit load at node 5-y
R	variation factors
αP	Due to applied loads

x, y denote the two coordinate directions

Equation (3.17)

$$\begin{bmatrix} \delta^*_{1x} \\ \delta^*_{1y} \\ \vdots \\ \delta^*_{16y} \end{bmatrix} = \begin{bmatrix} \delta_{1x} \\ \delta_{1y} \\ \vdots \\ \delta_{16y} \end{bmatrix} + \begin{bmatrix} \delta_{1x,1x} & \delta_{1x,1y} & \cdots & \delta_{1x,5y} \\ \delta_{1y,1x} & \delta_{1y,1y} & \cdots & \delta_{1y,5y} \\ \vdots & \vdots & \vdots & \vdots \\ \delta_{16y,1x} & \delta_{16y,1y} & \cdots & \delta_{16y,5y} \end{bmatrix} \begin{bmatrix} R_{c1} \\ R_{c2} \\ \vdots \\ R_{c8} \end{bmatrix}$$

Equation (3.19)

The displacements of the modified structure are then evaluated using equation (3.11). The matrix form of this equation is shown on page 56 (equation (3.19)). The eight columns of the matrix represent the displacements of the structure due to each of the unit loads. The vector $\{\delta_{1x}, \delta_{1y}, \ldots\ldots \delta_{16y}\}$ are the displacements due to applied loads. Similarly equation (3.12) may be written in matrix form. Finally equations (3.13) and (3.15) are used to calculate the strains and stresses of the modified structure.

The condensed variation factors were obtained from the summation of the variation factors. And the variation factors were evaluated from the solution of equation (3.3). One weakness of this technique is that a large matrix will result from equation (3.3). However it is possible to calculate the condensed variation factors directly from (3.10). This is more computationally efficient since the number of equations to be solved will be reduced. The formation of equation (3.10) may be derived from (3.3) and this is explained by again considering the structure shown in figure 3.1.

The modified part of the structure of figure 3.1, is shown in figure 3.2. There are 8 unit loads acting at nodes 1,2,5 and 6 as shown in the figure below.

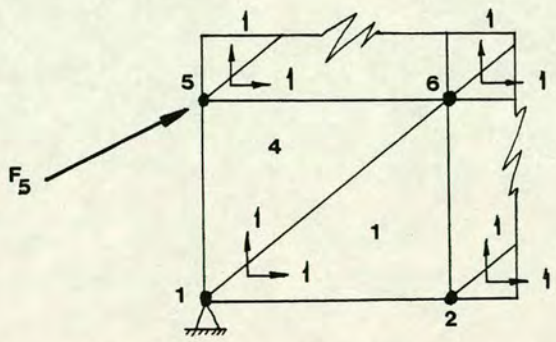


Figure 3.2 Part of the structure shown in figure 3.1

Each unit load analysis will give 6 nodal forces for each modified element. The nodal forces at common nodes 1 and 6 are different because elements 1 and 4 have different stiffnesses.

The matrix equation expressing equilibrium at the nodes as a result of the modification of elements 1 and 4 is equation (3.17). The first six rows of this matrix are the equilibrium equations for element

1 and the next six for element 4. The f_{ij}'s in column 1 and 7 are the same because node 1 is acted on by the same unit load and the same stiffness was used to evaluate it. Similarly for columns 2 and 8, 5 and 9, and finally 6 and 10. Writing the 1st. and 7th. rows of this matrix as:

$$(1+\alpha_1 f_{1,1})R_1 + \alpha_1 f_{1,2} R_2 + \ldots\ldots + \alpha_1 f_{1,12} R_{12} + \alpha_1 P_1 = 0 \qquad (3.20a)$$

and

$$\alpha_7 f_{7,1} R_1 + \alpha_7 f_{7,2} R_2 + \ldots\ldots + (1+\alpha_7) f_{7,7} R_7 + \ldots\ldots + \alpha_7 P_7 = 0 \qquad (3.20b)$$

Adding these together and noting that:

$$\begin{aligned} f_{1,1} &= f_{1,7} \;;\; f_{1,2} = f_{1,8} \;;\; f_{1,5} = f_{1,9} \;;\; f_{1,6} = f_{1,10} \\ f_{7,1} &= f_{7,7} \;;\; f_{7,2} = f_{7,8} \;;\; f_{7,5} = f_{7,9} \;;\; f_{7,6} = f_{7,10} \end{aligned} \qquad (3.21)$$

gives:

$$\begin{aligned} &(1+\alpha_1 f_{1,1} + \alpha_7 f_{7,1})(R_1 + R_7) + (\alpha_1 f_{1,2} + \alpha_7 f_{7,2})(R_2 + R_8) + \ldots\ldots \\ &+ (\alpha_1 f_{1,12} + \alpha_7 f_{7,12})R_{12} + (\alpha_1 P_1 + \alpha_7 P_7) = 0 \end{aligned} \qquad (3.22)$$

This summation may be undertaken for the 2nd. row with the 8th. row, 5th. row with the 9th. row and the 6th. row with the 10th. row. The resulting matrix is shown on the following page as equation (3.23). The variation and condensed variation factors are given by:

$$\{R\} = \{R_1 + R_7, R_2 + R_8, R_3, R_4, R_5 + R_9, R_6 + R_{10}, R_{11}, R_{12}\} \qquad (3.24a)$$

and is equivalent to:

$$\{R_c\} = \{R_{c1}, R_{c2}, R_{c3}, R_{c4}, R_{c5}, R_{c6}, R_{c7}, R_{c8}\} \qquad (3.24b)$$

and is the same as equation (3.18).

The obvious advantage is that the storage requirements are less and hence the solution takes less time. Equation (3.23) may be formed in a similar manner to that of the overall stiffness matrix. The contribution of the internal forces for each modified element may be added to form the overall matrix. Unfortunately the matrices of equations (3.17) and (3.23) are full and unsymmetric.

$$
\begin{array}{l}
\text{common node 1} \\
\\
\text{node 2} \\
\\
\text{common node 6} \\
\\
\text{node 5} \\
\\
\end{array}
\begin{bmatrix}
1+\alpha_1 f_{1,1}+\alpha_7 f_{7,1} & \alpha_1 f_{1,2}+\alpha_7 f_{7,2} & \cdots\cdots & \alpha_1 f_{1,12}+\alpha_7 f_{7,12} \\
\alpha_2 f_{2,1}+\alpha_8 f_{8,1} & 1+\alpha_2 f_{2,2}+\alpha_8 f_{8,2} & \cdots\cdots & \alpha_2 f_{2,12}+\alpha_8 f_{8,12} \\
\alpha_3 f_{3,1} & \alpha_3 f_{3,2} & \cdots\cdots & \alpha_3 f_{3,12} \\
\alpha_4 f_{4,1} & \alpha_4 f_{4,2} & \cdots\cdots & \alpha_4 f_{4,12} \\
\alpha_5 f_{5,1}+\alpha_9 f_{9,1} & \alpha_5 f_{5,2}+\alpha_9 f_{9,2} & \cdots\cdots & \alpha_5 f_{5,12}+\alpha_9 f_{9,12} \\
\alpha_6 f_{6,1}+\alpha_{10} f_{10,1} & \alpha_6 f_{6,2}+\alpha_{10} f_{10,2} & \cdots\cdots & \alpha_6 f_{6,12}+\alpha_{10} f_{10,12} \\
\alpha_{11} f_{11,1} & \alpha_{11} f_{11,2} & \cdots\cdots & \alpha_{11} f_{11,12} \\
\alpha_{12} f_{12,1} & \alpha_{12} f_{12,2} & \cdots\cdots & 1+\alpha_{12} f_{12,12}
\end{bmatrix}
\begin{bmatrix}
R_1+R_7 \\ R_2+R_8 \\ R_3 \\ R_4 \\ R_5+R_9 \\ R_6+R_{10} \\ R_{11} \\ R_{12}
\end{bmatrix}
+
\begin{bmatrix}
\alpha_1 P_1+\alpha_7 P_7 \\ \alpha_2 P_2+\alpha_8 P_8 \\ \alpha_3 P_3 \\ \alpha_4 P_4 \\ \alpha_5 P_5+\alpha_9 P_9 \\ \alpha_6 P_6+\alpha_{10} P_{10} \\ \alpha_{11} P_{11} \\ \alpha_{12} P_{12}
\end{bmatrix}
=
\begin{bmatrix}
0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0
\end{bmatrix}
$$

Due to unit load at node 1-x | Due to unit load at node 1-y | Due to unit load at node 5-y | Condensed variation factors | Due to applied loads

x, y denote the two coordinate directions

Equation (3.23)

3.2.2 Matrix forms of the theorems

The summation signs of equations (3.10),(3.11) and (3.12) serve as a useful purpose to indicate that the modified structure may be obtained by superposition of factored unit load analyses. They may be recast in matrix notation[12] which is particularly useful in computer implementation. For simplicity the effect of changing one element will be considered first. The matrix form of the equilibrium equations (3.10) are:

$$([I^e] + \lceil\alpha^e\rfloor[f^e])\{R^e\} + \lceil\alpha^e\rfloor\{P^e\} = \{0\} \tag{3.25}$$

where:

$[I^e]$ = identity matrix of e'th element;

$\lceil\alpha^e\rfloor$ = a diagonal matrix which express changes of the e'th element;

$[f^e]$ = internal forces of e'th element due to unit loads;

$\{R^e\}$ = variation factors of e'th element; and

$\{P^e\}$ = internal forces of e'th element due to applied loads.

The diagonal coefficients of $\lceil\alpha^e\rfloor$ are $\alpha_1,\alpha_2,.....\alpha_M$ from equation (3.2). The internal forces $[f^e]$ and $\{P^e\}$ are evaluated from the original element stiffness and nodal displacements:

$$[f^e \mid P^e] = [K^e][\delta_I^e \mid \delta^e] \tag{3.26}$$

where $[\delta_I^e \mid \delta^e]$ = element nodal displacements due to unit and applied loads.

The element nodal displacements are obtained from $[\delta_I \mid \delta]$.

If there are N elements to be modified then the equilibrium equations may be written:

$$([I] + \sum_{e=1}^{N} \overline{\lceil\alpha^e\rfloor}\,\overline{[f^e]})\sum_{e=1}^{N} \overline{\{R^e\}} + \sum_{e=1}^{N} \overline{\lceil\alpha^e\rfloor}\,\overline{\{P^e\}} = \{0\} \tag{3.27a}$$

or

$$([I] + \lceil\alpha\rfloor[f])\{R_c\} + \lceil\alpha\rfloor\{P\} = \{0\} \qquad (3.27b)$$

where:

$$\sum_{e=1}^{N} \lceil\overline{\alpha^e}\rfloor[\overline{f^e}] = \lceil\alpha\rfloor[f] \qquad (3.27c)$$

$$\sum_{e=1}^{N} \{\overline{R^e}\} = \{R_c\} \qquad (3.27d)$$

$$\sum_{e=1}^{N} \lceil\overline{\alpha^e}\rfloor\{\overline{P^e}\} = \lceil\alpha\rfloor\{P\} \qquad (3.27e)$$

and

$[I]$ = overall identity matrix;

$\lceil\alpha\rfloor$ = overall matrix of structural changes;

$[f]$ = overall internal force matrix due to unit loads; and

$\{P\}$ = overall internal force vector due to applied loads.

The bar sign(-) over the matrices and vectors in equation (3.27a) indicates that the matrices and vectors have been expanded to the total number of degrees of freedom for the modified parts of the structure. This is equal to W, given by equation (3.9). Obviously the expansion of $\lceil\overline{\alpha^e}\rfloor$, $[\overline{f^e}]$, $\{\overline{R^e}\}$ and $\{\overline{P^e}\}$ are not carried out in the computer implementation. Rather equation (3.25) is formed first for each modified element and then added into equation (3.27a) by correspondence of the local to global degrees of freedom. This is similar to the formation of the overall stiffness matrix.

The matrix form of equation (3.11) is:

$$\{\delta^*\} = \{\delta\} + [\delta_I]\{R_c\} \qquad (3.28)$$

and for equation (3.12):

$$\{T^*\} = \{T\} + [T_I]\{R_c\} \qquad (3.29)$$

Strains and stresses are evaluated from equations (3.13) and (3.15) respectively.

The form of equations (3.25) to (3.29) will be useful for comparing with the theorems of geometric variation.

3.3 The theorems of geometric variation

The method for the finite element formulation follows from the technique of trusses[26,139] which will be summarised first. If the joint coordinates of a structure are varied, then the member lengths and angles between the members connected to the joints are also varied. The latter variation is termed rotation since the clockwise angle of the member to the vertical axis changes. The former variation is termed elongation since the member changes in length.

Consider the coordinate of end j of the i'th member in figure 3.3(i) is varied as shown in figure 3.3(ii). The member undergoes rotation and elongation. The force in the modified member may be obtained, if first the original structure is analysed for unit loads applied at each degree of freedom at ends j and k.

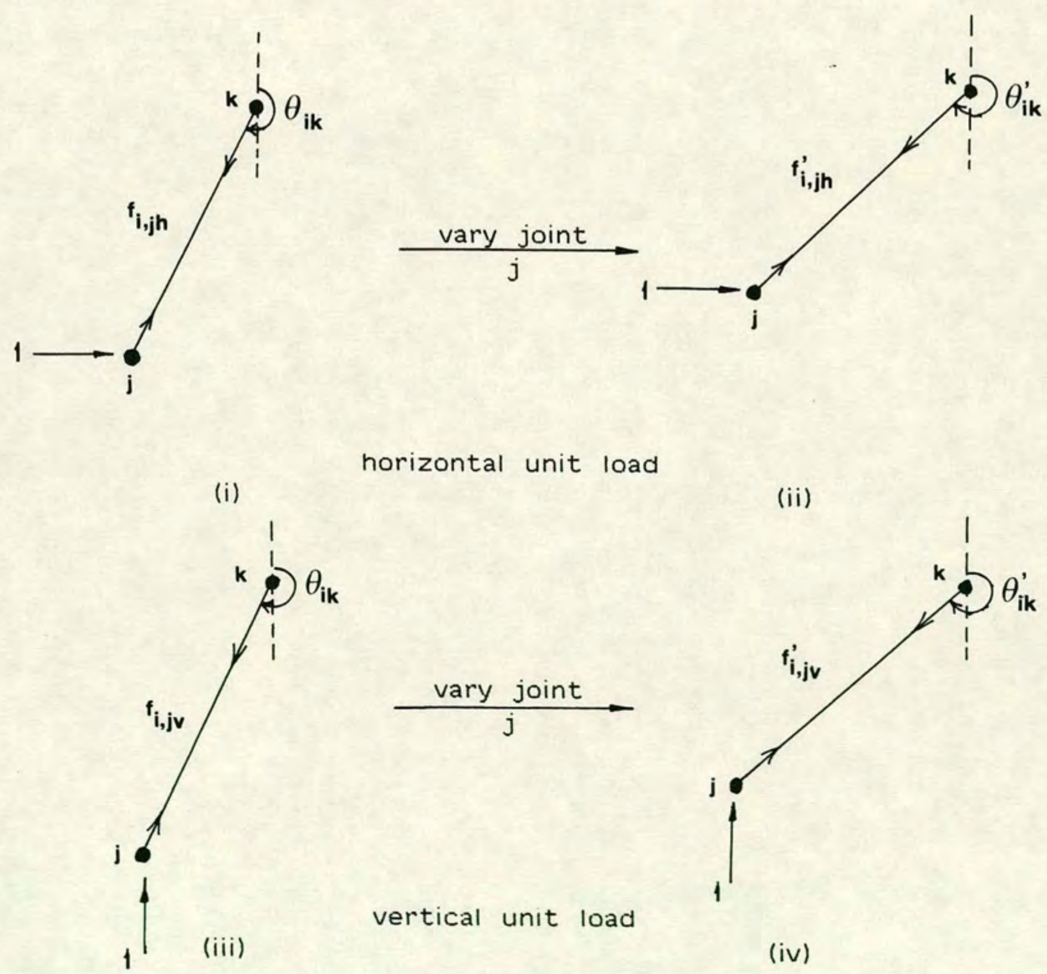


Figure 3.3

There are a total of four unit loads for each modified member as a result of a joint variation. The unit loads applied at joint j of the modified i'th member are shown before and after variation in figure 3.3.

For simplicity only unit loads acting at joint j will be considered first (for a complete assessment of the modification of member i, unit loads at joint k must also be considered). The member force due to the unit loads are $f_{i,jh}$ and $f_{i,jv}$ as shown in figures 3.3(i) and (iii). The subscripts i,jh and i,jv denote the force in the i'th member due to unit horizontal and vertical load at end j respectively. The force in the modified member in figure 3.3(ii) may be obtained by considering the stress-strain relationship:

$$f'_{i,jh} = E_i A_i \left[\frac{L_i^* - L_i'}{L_i'} \right] \tag{3.30a}$$

and

$$L_i' = \sqrt{\{(X_k - X_j)^2 + (Y_k - Y_j)^2\}} \tag{3.30b}$$

$$L_i^* = \sqrt{\{\{(X_k + \delta_{kh,jh}) - (X_j + \delta_{jh,jh})\}^2 + \{(Y_k + \delta_{kv,jh}) - (Y_j + \delta_{jv,jh})\}^2\}} \tag{3.30c}$$

where:

$f'_{i,jh}$ = internal force of i'th modified member due to unit horizontal load;

L_i' = length of i'th member after joint variation;

L_i^* = is termed the 'stretch' length assuming that the displacements are the same as before variation;

X_j, Y_j, X_k, Y_k = coordinates of joint j and k respectively after joint variation;

$\delta_{jh,jh}, \delta_{jv,jh}$ = displacements at joint j due to unit horizontal load at node j of original structure; and

$\delta_{kh,jh}, \delta_{kv,jh}$ = displacements at joint k due to unit horizontal load at node j of original structure.

A similar relationship also holds for the unit vertical load at end j.

However in addition the member suffers a change in rotation to θ'_{ik} from θ_{ik} due to the joint variation. The subscript ik denotes that the angle is measured at end k of the i'th member as shown in figures 3.3(ii) and (iv). The elongation and rotation will result in unbalanced forces and these are termed compensation forces. The compensation forces at joint k due to the unit horizontal load at joint j are:

$$C_{kh,jh} = f_{i,jh}\sin\theta_{ik} - f'_{i,jh}\sin\theta'_{ik} \tag{3.31a}$$

$$C_{kv,jh} = f_{i,jh}\cos\theta_{ik} - f'_{i,jh}\cos\theta'_{ik} \tag{3.31b}$$

where:

$C_{kh,jh}$ = horizontal compensation force at joint k, due to the modification of member i under a unit horizontal load applied at joint j;

$C_{kv,jh}$ = vertical compensation force at joint k, due to the modification of member i under a unit horizontal load applied at joint j;

θ_{ik} = original angle to the vertical at node k of i'th member; and

θ'_{ik} = modified angle to the vertical at node k of i'th member.

Similarly for the compensation forces at end j due to the unit horizontal load at joint j. The compensation forces for the other unit loads jv, kh and kv are evaluated using similar equations to (3.31). The various compensation forces for a member are tabulated in table 3.1.

		Unit load at joint j		Unit load at joint k	
Compensation forces		Horiz.	Vert.	Horiz.	Vert.
At joint j	Horiz.	$C_{jh,jh}$	$C_{jh,jv}$	$C_{jh,kh}$	$C_{jh,kv}$
	Vert.	$C_{jv,jh}$	$C_{jv,jv}$	$C_{jv,kh}$	$C_{jv,kv}$
At joint k	Horiz.	$C_{kh,jh}$	$C_{kh,jv}$	$C_{kh,kh}$	$C_{kh,kv}$
	Vert.	$C_{kv,jh}$	$C_{kv,jv}$	$C_{kv,kh}$	$C_{kv,kv}$

Table 3.1 Compensation forces of a member

From table 3.1 a total of 16 compensation forces must be evaluated using equations similar to that of (3.31).

So far the discussion has been based on a single modified member. If there are N members connected at joint k, whose joints are also varied these members will also sustain elongations and rotations. Therefore unit load analyses at the other ends of the N members are also required. The net compensation forces at a typical joint k is the sum of each member contributions. This is given by:

$$C_{kh,jh} = \sum_{i=1}^{N} (f_{i,jh} \sin\theta_{ik} - f'_{i,jh} \sin\theta'_{ik}) \tag{3.32a}$$

$$C_{kv,jh} = \sum_{i=1}^{N} (f_{i,jh} \cos\theta_{ik} - f'_{i,jh} \cos\theta'_{ik}) \tag{3.32b}$$

If the applied loads are at the nodes of the N modified members, the equilibrium equations may be formed. This may be undertaken by using the principle of superposition of unit load analyses and applied loads. The compensation forces are evaluated from the unit load analyses and if these are scaled the equilibrium equations are:

$$\begin{aligned}
&(1+C_{jh,jh})r_{jh} + C_{jh,jv}r_{jv} + \ldots + C_{jh,kv}r_{kv} = F_{jh} \\
&C_{jv,jh}r_{jh} + (1+C_{jv,jv})r_{jv} + \ldots + C_{jv,kv}r_{kv} = F_{jv} \\
&\cdot \\
&\cdot \\
&\cdot \\
&\cdot \\
&\cdot \\
&\cdot \\
&C_{kv,jh}r_{jh} + C_{kv,jv}r_{jv} + \ldots + (1+C_{kv,kv})r_{kv} = F_{kv}
\end{aligned} \tag{3.33}$$

where :

r_{jh} etc. = horizontal scale factor at joint j etc.; and

F_{jh} etc. = applied horizontal load at joint j etc.

The subscripts j and k denote the joints of the modified N members. These subscripts vary from j=1 to j=k, where k is now the total number of joints of the N modified members. Therefore the total number of equilibrium equations is equal to the number of joints of the N members. These joints are termed the affected joints.

To generalise the preceding concepts, consider the case when there are 'a' affected joints. The total number of affected joints is the sum of the varied joints together with all joints connected to the joints that are varied. Similarly the number of **members** affected is equal to the total number of members connected to the varied joints. At the 'a' affected joints there are applied loadings as in the example in equation (3.33). However there may be other joints, not included in 'a', that carry applied loads. These are termed the 'u' unaffected joints. To form the equilibrium equations, (a+u) multiplied by the degrees of freedom per joint, unit load analyses are required at each degree of freedom. For a plane truss this is equal to 2(a+u). Therefore by scaling the compensation forces at the (a+u) joints, the equilibrium equations in matrix form are:

$$\left(\left[\begin{array}{c|c} I & 0 \\ \hline 0 & I \end{array}\right] + \left[\begin{array}{c|c} C_{aa} & C_{au} \\ \hline C_{ua} & C_{uu} \end{array}\right] \right) \left[\begin{array}{c} r_a \\ \hline r_u \end{array}\right] = \left[\begin{array}{c} F_a \\ \hline F_u \end{array}\right] \qquad (3.34)$$

where:

$[C_{aa}]$ = matrix of compensation forces at 'a' joints due to unit loadings at 'a' joints;

$[C_{au}]$ = matrix of compensation forces at 'a' joints due to unit loadings at 'u' joints;

$[C_{ua}]$ = matrix of compensation forces at 'u' joints due to unit loadings at 'a' joints;'

$[C_{uu}]$ = matrix of compensation forces at 'u' joints due to unit loadings at 'u' joints;

$\{r_a\}$ = scale factors for 'a' joints;

$\{r_u\}$ = scale factors for 'u' joints;

$\{F_a\}$ = applied loads at 'a' joints; and

$\{F_u\}$ = applied loads at 'u' joints.

If there is no applied loading at the 'u' joints then equation (3.34) reduces to equation (3.33).

The compensation forces at the 'u' joints are zero, since they are not involved in any changes. Therefore:

$$[C_{ua}] = [0] \quad ; \quad [C_{uu}] = [0] \tag{3.35a,b}$$

Substitution of equation (3.35) into (3.34) and considering only the lower half gives:

$$\{r_u\} = \{F_u\} \tag{3.36}$$

Substitution of (3.36) into the upper half of (3.34) gives:

$$\{r_a\} = ([I]+[C_{aa}])^{-1}(\{F_a\}-[C_{au}]\{r_u\}) \tag{3.37}$$

When the scale factors have been solved, the member forces and displacements of the modified structure are obtained as follows:

Forces in the modified members may be calculated using:

$$P_i^* = \sum_{j=1}^{a+u}(r_{jh} f'_{i,jh} + r_{jv} f'_{i,jv}) \tag{3.38}$$

and for the unaltered members:

$$P_i^* = \sum_{j=1}^{a+u}(r_{jh} f_{i,jh} + r_{jv} f_{i,jv}) \tag{3.39}$$

Equations (3.38) and (3.39) are the first theorem of geometric variation, and the second theorem is given by the modified displacements (for each component) which are:

$$\delta^* = \sum_{j=1}^{a+u}(r_{jh} \delta_{jh} + r_{jv} \delta_{jv}) \tag{3.40}$$

where:

δ_{jh} = displacement due to unit horizontal load at node j; and

δ_{jv} = displacement due to unit vertical load at node j.

The matrix form of the theorems of geometric variation will be given in section 3.3.2.

3.3.1 Extension to finite element problems

The theorems of geometric variation may be extended to finite element problems. The formulations in section 3.3 were based on intuitive and physical reasoning. Drawing the analogy with truss structures, the edge forces of a 3-node triangular element may be evaluated. This triangular element is called the natural stiffness element first developed by Argyris[2,5,Appendix III]. The state of stress in the element is defined by edge extensions in place of the nodal displacements. The element stiffness matrix is (3 by 3) instead of the usual (6 by 6) for a constant strain triangular element.

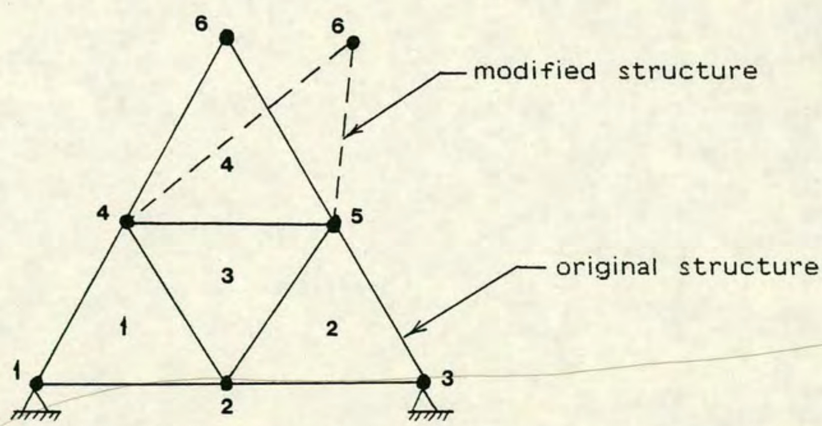


Figure 3.4

The generalisation of the theorems of geometric variation to finite element problems may be considered by using the idealisation shown in figure 3.4 which is assembled from four natural stiffness elements. If node 6 is changed then the affected nodes 'a' are 4,5 and 6. By analogy with the truss structures the element edges (4-5,6-5 and 4-6) may be treated as members. Similarly the edge extensions and forces are member extensions and forces. The elemental edge forces of the original structure may be determined from:

$$\{f^e\}_{jh} = [K_N^e]\{d^e\}_{jh} \tag{3.41}$$

where:

$\{f^e\}_{jh}$ = edge forces of an element due to unit horizontal load at node j;

$[K_N^e]$ = natural stiffness of original element; and

$\{d^e\}_{jh}$ = edge extensions of an element due to unit horizontal load at node j.

A similar equation to (3.41) may be written for the unit vertical load. The node j is usually one of the (a+u) nodes.

Assuming that the edge extensions remain the same before and after modification, then the edge forces of the modified element are:

$$\{f'^e\}_{jh} = [K_N^{*e}]\{d^e\}_{jh} \tag{3.42}$$

where:

$\{f'^e\}_{jh}$ = edge forces of modified element due to unit horizontal load at node j of original structure; and

$[K_N^{*e}]$ = natural stiffness of modified element.

By comparing equations (3.30a) with (3.42), it is obvious that both are similar. The term E_iA_i/L'_i in (3.30a) corresponds to $[K_N^{*e}]$. Similarly the member extensions $(L_i^* - L'_i)$ corresponds to the edge extensions $\{d^e\}$. Equation (3.30a) is a special case of (3.42) since truss members are one-dimensional finite elements. By the same reasoning, the compensation forces at the affected nodes, with N edges connected to it are given by equation (3.32). Hence the matrix of compensation forces (3.34) may be formed. The analysis then proceeds using equations (3.38),(3.39) and (3.40) as before.

However it should be noted that the number of element edges connected to a node is different from that of trusses.

Consider figure 3.4 again with node 6 varied. If figure 3.4 is a truss structure, then the affected nodes are 4,5 and 6. Compensation forces are then evaluated at nodes 4,5 and 6 due to unit loads acting at these nodes (if the applied loading is at node 2, unit loads at this node must also be considered). At node 4 (or 5), the force in member 4-5 may be excluded in the calculation of compensation forces, as this member does not suffer elongation or rotation.

On the otherhand, if figure 3.4 is a finite element structure, and using the concept of edge forces, then edge 4-5 of element 4 must be included, but edge 4-5 of element 3 is excluded in the evaluation of compensation forces. This is because the stiffness of element 4 is modified (but not that of element 3), although edge 4-5 does not suffer elongation or rotation. Edge 4-5 may be visualised as two members connecting node 4 to 5. One member has a stiffness due to element 4 which is modified and the other stiffness is due to the unmodified element 3.

3.3.2 Generalisation of the theorems of geometric variation

Although the natural stiffness element serves as a useful analogy with truss structures, the concept of edge forces is not directly applicable to other elements. An important point about equation (3.32) is that the edge forces must be resolved at the nodes. These resolved forces are actually internal nodal forces. Therefore considering the (6 by 6) triangular element stiffness matrix, an equation similar to (3.41) may be written as:

$$\{f^e\}_{jh} = [K^e]\{\delta^e\}_{jh} \tag{3.43}$$

where:

$\{f^e\}_{jh}$ = nodal forces of an element due to unit horizontal load at node j;

$[K^e]$ = (6 by 6) element stiffness matrix;

$\{\delta^e\}_{jh}$ = nodal displacements of an element due to unit horizontal load at node j;

and for equation (3.42):

$$\{f'^{e}\}_{jh} = [K^{*e}]\{\delta^{e}\}_{jh} \tag{3.44}$$

where:

$\{f'^{e}\}_{jh}$ = nodal forces of modified element due to unit horizontal load at node j; and

$[K^{*e}]$ = (6 by 6) modified element stiffness matrix.

These two equations are also applicable to the unit vertical load.

Due to the resolution of the edge forces at the nodes then:

$$f_{i,jh}\sin\theta_{ik} = -\,f^{e}_{ih,jh} \quad ; \quad f'_{i,jh}\sin\theta'_{ik} = -\,f'^{e}_{ih,jh} \tag{3.45a}$$

$$f_{i,jh}\cos\theta_{ik} = -\,f^{e}_{iv,jh} \quad ; \quad f'_{i,jh}\cos\theta'_{ik} = -\,f'^{e}_{iv,jh} \tag{3.45b}$$

The forces $f^{e}_{ih,jh}, f^{e}_{iv,jh}, f'^{e}_{ih,jv}$ and $f'^{e}_{iv,jv}$ are internal nodal forces in the horizontal and vertical directions respectively due to unit horizontal load at node j. These forces are obtained from the vectors $\{f^{e}\}_{jh}$ and $\{f'^{e}\}_{jh}$ of equations (3.43) and (3.44) respectively. And similarly for the unit vertical load. Hence equations (3.32a) and (3.32b) become:

$$C_{kh,jh} = \sum_{i=1}^{N}(f'^{e}_{ih,jh} - f^{e}_{ih,jh}) \tag{3.46a}$$

$$C_{kv,jh} = \sum_{i=1}^{N}(f'^{e}_{iv,jh} - f^{e}_{iv,jh}) \tag{3.46b}$$

The compensation forces from equation (3.46) are the out of balance forces at the nodes as a result of changes in the stiffness of the elements connected to the node. Equations (3.43),(3.44) and (3.46) may also be applied to other elements. The use of changes in elongation and orientation which may lead to confusion is avoided. Instead compensation forces are obtained from the difference of the modified and original element stiffnesses multiplied by the displacements of the original structure due to unit loads. For efficient computer implementation the theorems of geometric variation in matrix form are now outlined.

3.3.2.1 Initial analysis

To illustrate the general procedure for reanalysis using the theorems of geometric variation, the finite element structure of figure 3.1 will be considered. The response of the modified structure when the coordinate of node 16 is varied will be determined. The variation of node 16 will affect the stiffnesses of elements 15 and 18. The affected nodes, 'a' are 11,12,15 and 16. In addition there exist applied loads at nodes 5 and 9. These are the 'u' nodes and they are not included in 'a'. The original structure is first analysed for unit loads at the (a+u) nodes for each degree of freedom. This analysis is given by:

$$[K][\delta_a \mid \delta_u] = [I_a \mid I_u] \tag{3.47}$$

where:

$[\delta_a \mid \delta_u]$ = displacements due to unit loadings at 'a' and 'u' nodes; and

$[I_a \mid I_u]$ = unit loads at 'a' and 'u' nodes.

The matrix $[I_a \mid I_u]$ contains zeros except for those coefficients corresponding to the (a+u) nodes which will be equal to one. The number of columns of the submatrix $[I_a]$ for figure 3.1 is 2a = 8 and for submatrix $[I_u]$ it is 2u = 4. The factor two is required because each node has two degrees of freedom. Note that there is no need to analyse for the applied loadings, because the results may easily be obtained by superposition of the unit load analyses.

The reactions at the boundary nodes may be calculated at the same time as the solution of equation (3.47). These reactions are denoted by:

$$[T_a \mid T_u] \tag{3.48}$$

where the subscripts 'a' and 'u' refer to reactions at boundary nodes due to unit loads at (a+u) nodes.

3.3.2.2 Analysis of modified structure

After the initial analysis has been performed on the original structure, the response of the modified structure may be obtained. As mentioned earlier, the effect of varying the coordinates of the nodes, modifies the elements connected to the nodes. The stiffness of the original element, $[K^e]$ changes to a new stiffness $[K^{*e}]$. If the number of modified elements is N the element nodal forces are given by:

$$[f_a^e \mid f_u^e] = [K^e][\delta_a^e \mid \delta_u^e] \tag{3.49a}$$

$$[f_a'^e \mid f_u'^e] = [K^{*e}][\delta_a^e \mid \delta_u^e] \tag{3.49b}$$

where:

$[f_a^e \mid f_u^e]$ = nodal forces due to unit loads at (a+u) nodes before the element is modified; and

$[f_a'^e \mid f_u'^e]$ = nodal forces due to unit loads at (a+u) nodes after the element is modified.

The element nodal displacements are obtained from $[\delta_a \mid \delta_u]$. The modified element stiffness, in equation (3.49b), may be determined using the expression:

$$[K^{*e}] = \int_{\Omega_e^*} [B^*]^T [D][B^*] \, d\Omega_e^* \tag{3.49c}$$

where $[B^*]$ = modified strain-displacement matrix.

The same displacements are used in equations (3.49a) and (3.49b). The forces in the modified structure will differ from that of the original one because of the change in element stiffness. The compensation forces are given by the difference between these two sets of forces. Therefore from equations (3.49a) and (3.49b) for each element:

$$\begin{aligned}
[C_a^e \mid C_u^e] &= [f_a'^e \mid f_u'^e] - [f_a^e \mid f_u^e] \\
&= ([K^{*e}]-[K^e])[\delta_a^e \mid \delta_u^e] \\
&= [\Delta K^e][\delta_a^e \mid \delta_u^e]
\end{aligned} \tag{3.50}$$

where:

$[C^e_a \mid C^e_u]$ = matrix of compensation forces due to unit load analyses at (a+u) nodes; and

$[\Delta K^e]$ = change in stiffness matrix of e'th element.

Equation (3.50) is similar in form to equation (3.34) for a truss member. Here the submatrix $[C^e_a]$ is equal to $[C^e_{aa}]$ and $[C^e_u]$ is equal to $[C^e_{au}]$ because compensation forces only exist at the nodes of the modified elements. The equilibrium equations for one element is similar to that of (3.34) and is:

$$\left(\left[\begin{array}{c|c} I^e & 0 \\ \hline 0 & I^e \end{array}\right] + \left[\begin{array}{c|c} C^e_{aa} & C^e_{au} \\ \hline 0 & 0 \end{array}\right]\right)\left[\begin{array}{c} r^e_a \\ \hline r^e_u \end{array}\right] = \left[\begin{array}{c} F^e_a \\ \hline F^e_u \end{array}\right] \tag{3.51}$$

where $[C^e_{ua}]$ and $[C^e_{uu}]$ are both zeros. The superscript e denotes e'th modified element.

Therefore for N modified elements the equilibrium equations are:

$$\left(\left[\begin{array}{c|c} I & 0 \\ \hline 0 & I \end{array}\right] + \sum_{e=1}^{N}\left[\begin{array}{c|c} \bar{C}^e_{aa} & \bar{C}^e_{au} \\ \hline 0 & 0 \end{array}\right]\right)\sum_{e=1}^{N}\left[\begin{array}{c} \bar{r}^e_a \\ \hline \bar{r}^e_u \end{array}\right] = \sum_{e=1}^{N}\left[\begin{array}{c} \bar{F}^e_a \\ \hline \bar{F}^e_u \end{array}\right] \tag{3.52a}$$

or

$$\left(\left[\begin{array}{c|c} I & 0 \\ \hline 0 & I \end{array}\right] + \left[\begin{array}{c|c} C_{aa} & C_{au} \\ \hline 0 & 0 \end{array}\right]\right)\left[\begin{array}{c} r_a \\ \hline r_u \end{array}\right] = \left[\begin{array}{c} F_a \\ \hline F_u \end{array}\right] \tag{3.52b}$$

where:

$$\sum_{e=1}^{N}[\bar{C}^e_{aa}] = [C_{aa}] \tag{3.52c}$$

$$\sum_{e=1}^{N}[\bar{C}^e_{au}] = [C_{au}] \tag{3.52d}$$

$$\sum_{e=1}^{N}\{\bar{r}^e_a\} = \{r_a\} \tag{3.52e}$$

$$\sum_{e=1}^{N}\{\bar{r}^e_u\} = \{r_u\} \tag{3.52f}$$

$$\sum_{e=1}^{N}\{\bar{F}^e_a\} = \{F_a\} \tag{3.52g}$$

$$\sum_{e=1}^{N}\{\bar{F}^e_u\} = \{F_u\} \tag{3.52h}$$

The matrices and vectors, $[C_{aa}],[C_{au}],\{r_a\},\{r_u\},\{F_a\}$ and $\{F_u\}$ are the overall matrices and vectors. The bar sign(-) in equation (3.52a) indicates that the matrices and vectors have all been expanded to the total number of degrees of freedom of the affected elements. In the computer implementation the element compensation forces are first evaluated using equation (3.50) and then added to the overall compensation forces in equation (3.52a). This is similar to forming the overall stiffness matrix of the structure.

The modified displacements are:

$$\{\delta^*\} = [\delta_a \mid \delta_u] \begin{bmatrix} r_a \\ \hline r_u \end{bmatrix} \tag{3.53}$$

and the modified reactions are:

$$\{T^*\} = [T_a \mid T_u] \begin{bmatrix} r_a \\ \hline r_u \end{bmatrix} \tag{3.54}$$

The strains of the unmodified elements are:

$$\{\varepsilon^*\} = [B]\{\delta^*\} \tag{3.55a}$$

and for the modified elements:

$$\{\varepsilon^*\} = [B^*]\{\delta^*\} \tag{3.55b}$$

The stresses of the unmodified elements are:

$$\{\sigma^*\} = [D][B]\{\delta^*\} \tag{3.56a}$$

and for the modified elements:

$$\{\sigma^*\} = [D][B^*]\{\delta^*\} \tag{3.56b}$$

The modified internal forces (or the first theorem of geometric variation) are not required in finite element analysis.

3.3.2.3 An example

The example of figure 3.1 may be summarised as follows:

No. of affected elements N=2 (elements 15 and 18)

No. of affected nodes $a=4$ (nodes 11,12,15 and 16)

No. of unaffected nodes with applied loads $u=2$ (nodes 5 and 9)

Therefore no. of unit load analyses = $2(a+u) = 12$

Solution of equation (3.47) gives $[\delta_a \mid \delta_u]$ and $[T_a \mid T_u]$ may be determined at the same time. For each affected or modified element equation (3.50) is evaluated and added to the overall compensation matrix (3.52a). This matrix is written in full on the following page as equation (3.57). Each coefficient C_{ij} of the matrix is the compensation force at node i due to unit load at node j. The diagonal coefficients are of the form $(1+C_{ii})$. The first 'a' rows of the matrix in equation (3.57) are full and the matrix is unsymmetric. The scale factors $\{r_1, r_2, \ldots\ldots, r_{12}\}$ may be determined and hence the analysis of the modified structure may be obtained using equations (3.53),(3.54),(3.55) and (3.56).

The technique as outlined above is much easier to understand than using the concepts of elongation and orientation. The most important step of the calculation is the evaluation of the compensation forces and scale factors. This is simply given by equation (3.50) and is applicable to all types of elements.

3.4 A comparison between the theorems of structural and geometric variation

Both theorems result in a system of equilibrium equations which arise due to changes in the stiffness of a structure.

The main difference is that the theorems of structural variation can only predict the response of the modified structure due to structural changes. The theorems of geometric variation are more general, they may take into account both changes in structural properties and the geometry. However extra unit load analyses are required for the unaffected nodes. These are the nodes which are subject to applied loading but are not included with the affected nodes. The second difference is that the equilibrium equations of the theorems of structural variation are expressed in terms of the internal

$$
\underbrace{\left[\begin{array}{cccc:ccc}
1+C_{1,1} & C_{1,2} & \cdots & C_{1,8} & C_{1,9} & \cdots & C_{1,12} \\
C_{2,1} & 1+C_{2,2} & \cdots & C_{2,8} & C_{2,9} & \cdots & C_{2,12} \\
\vdots & \vdots & & \vdots & \vdots & & \vdots \\
C_{8,1} & C_{8,2} & \cdots & 1+C_{8,8} & C_{8,9} & \cdots & C_{8,12} \\
\hdashline
 & & O & & & I & \\
\end{array}\right]}_{\text{Due to unit loads at 'a' nodes} \quad | \quad \text{Due to unit loads at 'u' nodes}}
\underbrace{\left[\begin{array}{c}
r_1 \\ r_2 \\ \vdots \\ r_8 \\ \hdashline r_9 \\ \vdots \\ r_{12}
\end{array}\right]}_{\text{Scale factors}}
=
\underbrace{\left[\begin{array}{c}
0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ F_{16,x} \\ F_{16,y} \\ \hdashline F_{5,x} \\ F_{5,y} \\ F_{9,x} \\ F_{9,y}
\end{array}\right]}_{\text{Applied loads}}
\begin{array}{l}
\text{Node 11} \\ \text{Node 12} \\ \text{Node 15} \\ \text{Node 16} \\ \text{Node 5} \\ \text{Node 9}
\end{array}
\begin{array}{l}
\text{'a' nodes} \\ \text{'u' nodes}
\end{array}
$$

x ,y denote the two coordinate directions

Equation (3.57)

forces due to applied loads and internal unit load analyses. The theorems of geometric variation are expressed in terms of the externally applied loads and unit load analyses. This is advantageous since the magnitude and direction of the applied loads may change when the structure is modified.

The required number of unit load analyses is in general different. The theorems of geometric variation require extra unit load analyse at the 'u' nodes. If 'u' is zero then the number of unit load analyses required is the same for both theorems provided that the number of modified elements is the same. Further with the theorems of geometric variation, an analysis for the applied loading is not required.

The matrix $\lceil \alpha^e \rfloor [f^e]$ in equation (3.25) and the $[C^e_{aa}]$ in (3.51) are equivalent. This may easily be demonstrated by considering changing an element by an amount α. The new element stiffness matrix using the theorems of structural variation from equation (1.18) is:

$$[K^{*e}] = (1+\alpha^e)[K^e] \tag{3.58}$$

The new element stiffness is simply a scalar multiple of the original stiffness. Each column of $[f^e]$ is formed by considering unit load analyses at the nodes of the modified elements which are the 'a' affected nodes. These are evaluated using equation (3.26):

$$[f^e] = [K^e][\delta^e_I] \tag{3.59}$$

and multiplying by the matrix $\lceil \alpha^e \rfloor$:

$$\lceil \alpha^e \rfloor [f^e] = \lceil \alpha^e \rfloor [K^e][\delta^e_I] \tag{3.60}$$

From equation (3.58):

$$\alpha^e [K^e] = [K^{*e}] - [K^e] \tag{3.61}$$

Substitution of (3.61) into (3.60) and noting that:

$$\alpha^e [K^e] \equiv \lceil \alpha^e \rfloor [K^e] \tag{3.62}$$

gives:

$$\lceil \alpha^{e} \rfloor [f^{e}] = ([K^{*e}]-[K^{e}])[\delta^{e}_{I}] \tag{3.63}$$

But from equation (3.50):

$$[C^{e}_{a}] = ([K^{*e}]-[K^{e}])[\delta^{e}_{a}] \equiv [C^{e}_{aa}] \tag{3.64}$$

The affected nodes 'a' are also the nodes of the modified elements, and therefore the displacements due to the unit loads $[\delta^{e}_{I}]$ and $[\delta^{e}_{a}]$ are the same.

Hence by comparing equations (3.63) with (3.64):

$$[C^{e}_{aa}] \equiv \lceil \alpha^{e} \rfloor [f^{e}] \tag{3.65}$$

The main restriction on the theorems of structural variation is in equation (3.58) where the modified stiffness is a scalar multiple of the original stiffness. This is possible because only one parameter is changing, in this case the elastic modulus. When the geometry of the element changes this simple relation is not valid. The modified element stiffness must be formed from equation (3.49c) as required by the theorems of geometric variation. Therefore the effects of changes in element structural properties and geometry may be evaluated at the same time but at extra expense. In general, changes in the element will not result in a new element stiffnesses which are a scalar multiple of the original stiffness.

If the number of modified elements are the same for both theorems, then the solution of the same number of equilibrium equations is required. This means that the overall matrices $\lceil \alpha \rfloor [f]$ and $[C_{aa}]$ are equal in size. Therefore the effort needed to solve for the condensed variation factors from equation (3.27b) and the scale factors from equations (3.52b) and (3.37) will be the same. This is the most inefficient part of the reanalysis technique using either the theorems of structural or geometric variation, because the matrices are full and unsymmetric.

3.5 Summary

The theorems of structural and geometric variation have been derived for linear finite element analysis. The matrix forms and procedures outlined are particularly important for efficient computer implementation. At the same time both theorems have been generalised to handle various types of finite elements. Therefore a single computer program may be coded with different types of elements for used with this reanalysis technique.

The equilibrium equations are formulated in terms of the modified elements only. This is a reduced set of equations compared to the overall stiffness equations. If the number of modified elements are small, the reduced set of equations may be solved efficiently and hence the response of the modified structure may be obtained at less computational expense. However the reduced set of equations is full and unsymmetric. Ideally structures considered using these theorems should be large with only small areas of modification. The efficiency of both theorems and their potential applications are investigated in Chapters 5 and 6.

CHAPTER 4

THE THEOREMS OF STRUCTURAL AND GEOMETRIC VARIATION FOR NONLINEAR FINITE ELEMENT ANALYSIS

4.1 Introduction

In the previous chapter, the theorems were formulated for linear finite element analysis. The extension to nonlinear analysis requires repeated linear analysis of the nonlinear equilibrium equations to enable the formulations presented in Chapter 3 to be readily used.

Both the theorems of structural and geometric variation are proposed as solution techniques for nonlinear analysis of finite element structures. These theorems may be incorporated into the familiar Newton-Raphson methods. In this way the efficiency and accuracy of the results using the theorems may be assessed by comparison with those obtained using the usual formulation of the Newton-Raphson methods. In addition the limitations and difficulties of using the theorems of structural variation in nonlinear finite element analysis ~~is~~ are discussed. The theorems of geometric variation on the otherhand may readily be applied with minor modifications to the derivations of Chapter 3.

4.2 Newton-Raphson methods

A number of existing Newton-Raphson methods have been presented in Chapter 2. Essentially these methods involve applying the loads in increments with iterations within each load increment. During an iteration, the residual forces, which measure the departure of the solution from equilibrium are treated as applied loads to obtain the increments in displacements. This will ensure that equilibrium is maintained at the end of each load increment. Although the residual forces may be small at the end of a load increment, they may accumulate if a large number of load increments is used. Therefore to satisfy equilibrium throughout the analysis the residual forces at the end of a load increment are added to next load increment. Both measures will prevent the 'drifting' of the load-displacement curve. This approach is implemented in all the proposed techniques although it has not been explicitly mentioned in every one. Further it is assumed that the load increments are obtained by applying a single load factor to the total loads. This is known as proportional loading and although it is rather crude it serves to present the proposed techniques as an efficient nonlinear solution algorithm.

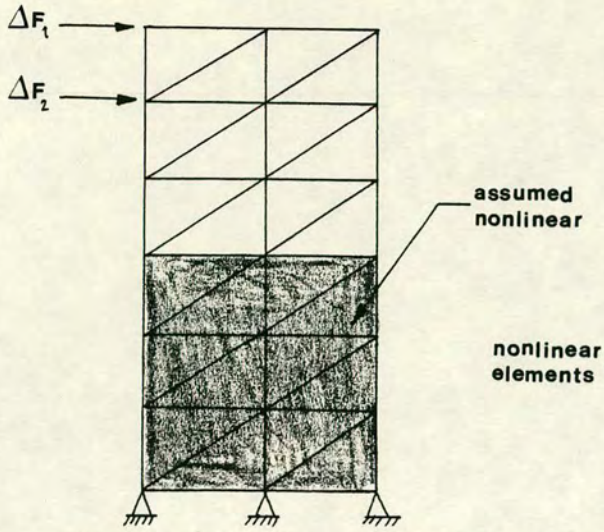
The formulation of the theorems using the Newton-Raphson procedures were the first attempt to use them as a nonlinear analysis technique in this way. They were first applied, to simple one-dimensional finite elements[Appendix IV] but later found not to be applicable to more general continuum problems. This is explained in section 4.2.1.4 and as a result the theorems of structural variation were abandoned as a nonlinear analysis technique.

The theorems of geometric variation present no difficulties as a nonlinear analysis technique. The formulations in Chapter 3 may easily be adapted to treat material and geometrical nonlinear problems. Hence they will be the technique used in Chapters 7,8 and 9 to investigate its efficiency.

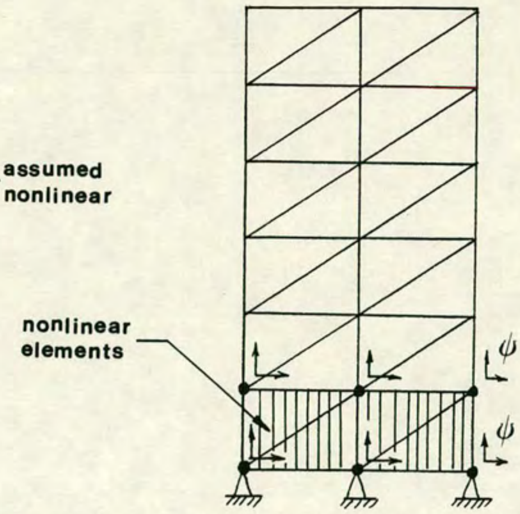
4.2.1 The theorems of structural variation

In nonlinear analysis it is highly likely that the stiffness of more than one element will vary. The equations required to take into account simultaneous changes in more than one element have been derived in Chapter 3. The procedure of simultaneous modifications is particularly important since the proposed strategies for nonlinear analysis will use these derivations. These strategies are equally applicable to material and geometrical nonlinear problems.

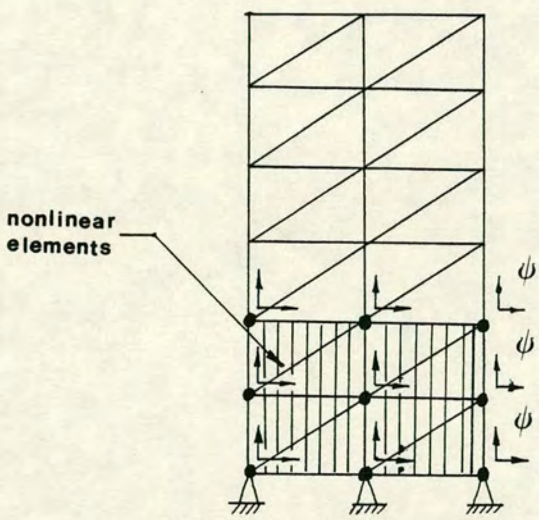
For example if the nonlinear response of the finite element structure shown in figure 4.1(i), with the applied loads, is to be analysed then the theorems will be efficient if the number of unit loads is small. Therefore the likely nonlinear elements should be known in advance and must be localised in nature. This area of nonlinearity is shown shaded in figure 4.1(i). Predicting the part of the structure likely to be nonlinear requires experience and physical understanding of the problem. If the assumed area of nonlinearity is too large then the method will become inefficient because the number of unit loads will also be large.



(i) First load increment and first iteration



(ii) At second iteration



(iii) At third iteration

Figure 4.1 Finite element structure

The initial analysis of the structure is performed for unit loads at the nodes of the shaded elements in addition to the applied loads. This analysis is given by:

$$[K_0][\delta_I \mid \delta] = [I \mid F] \tag{4.1}$$

Once the initial analysis has been performed the response of the modified structure may now be obtained.

4.2.1.1 The initial stiffness technique

The recursion equations for this technique by the Newton-Raphson method has already been derived in Chapter 2. In this technique the same linear stiffness matrix, $[K_0]$ is used throughout the analysis.

The use of the theorems of structural variation involves the superposition of applied and unit load analyses when the stiffness of the elements are modified. The unit load analyses are factored by condensed variation factors and the equilibrium equations at the nodes are solved to give the values of these factors. In the initial stiffness technique the diagonal matrix $\lceil \alpha \rfloor$ which defines changes in the element stiffnesses is a null matrix because the stiffnesses remain constant throughout the analysis. The only changing variables during the iteration are the residual forces. In this formulation the residual forces are the out of balance forces to be equilibrated by the unit load analyses. The residual forces are the condensed variation factors required to ensure equilibrium. The recursion equations within a load increment $\{\Delta F\}$ using the theorems of structural variation are therefore given by:

$$\{R_c^r\} = -\{\psi^r\} \tag{4.2a}$$

$$\{\delta^r\} = \{\delta^{r-1}\} + [\delta_I]\{R_c^r\} \tag{4.2b}$$

$$\{\psi^r\} = \int_\Omega [B^{r-1}]^T\{\sigma^{r-1}\}\, d\Omega - \{F\} \tag{4.2c}$$

As the elements become nonlinear the residual forces at the nodes of these elements are evaluated using equation (4.2c). These residual forces only act at the nodes of the nonlinear elements as shown in figures 4.1(ii) and (iii). At the next load increment the same set of equations (4.2) are used again. Note that if the applied loads are multiplied by a load factor, $\lambda\{F\}$ to obtain the load increment, $\{\Delta F\}$ (see equation (4.16)) then the displacements due to the applied loads, $\{\delta^{r-1}\}$ in equations (4.2b) and (4.6b) must also be multiplied by the same load factor at the first iteration of the load increment.

It is immediately obvious that the residual forces, $\{\psi^r\}$ are the condensed variation factors by studying the use of equation (4.2b) where the incremental displacements, $\{\Delta\delta^r\}$ are given by $[\delta_I]\{R^r_c\}$. This is simply an application of the principle of superposition in which the response of the modified structure is evaluated from the responses of factored unit load analyses. The internal forces of the modified structure are not required. The modified strains and stresses from Chapter 3 are respectively given by:

$$\{\varepsilon^r\} = [B]\{\delta^r\} \tag{4.3}$$

$$\{\sigma^r\} = [D][B]\{\delta^r\} \tag{4.4}$$

In equations (4.2c),(4.3) and (4.4) the matrices [D] and [B] are not constants. Depending on the sources of nonlinearities one or both of these matrices should be updated after each iteration. In the case of material nonlinearities, the matrix [D] is replaced by the elasto-plastic matrix $[D_{ep}]$. For geometric nonlinearities the [B] matrix must take into account the large displacements. Either of these nonlinearities result in residual forces at the nodes of the nonlinear elements. The residual forces are evaluated using equation (4.2c). Convergence is deemed to occur when the residual forces or incremental displacements have become tolerably small.

4.2.1.2 The tangential stiffness technique

In this technique the residual forces and the structure stiffness matrix are reformed and reduced every iteration. Hence there are two changing variables; the element stiffnesses and the residual forces at the nodes. The residual forces are again the condensed variation factors. Since the stiffness is changing the unit load analyses must be updated at each iteration. This may be accomplished by treating the unit loads as applied loads.

The first stage of the analysis cycle involves the modification of the unit load analyses. The recursion equations are given by the equilibrium conditions:

$$([I] + \lceil\alpha^r\rfloor[f])[R^r_{cI}] + \lceil\alpha^r\rfloor[f] = [0] \tag{4.5a}$$

and

$$[\delta^r_I] = [\delta_I] + [\delta_I][R^r_{cI}] \tag{4.5b}$$

where $[R^r_{cI}]$ = matrix of condensed variation factors for the unit load analyses.

Equation (4.5a) is a modified form of equation (3.27b) for multiple unit load analyses. The coefficients of the matrix [f] are the internal forces due to the unit loads. These coefficients are treated as if they are from the applied loads. A typical coefficient of the matrix of condensed variation factors is $R_{cI(ij)}$. It is the condensed variation factor for the i'th degree of freedom due to unit load acting at the j'th degree of freedom of the modified element. The columns of the matrix of condensed variation factors, $[R^r_{cI}]$ correspond to the condensed variation factors for each unit load. Equation (4.5a) is solved for $[R^r_{cI}]$ and the updated displacements due to the unit load analyses are evaluated using equation (4.5b). The modified internal forces due to the unit loads are not required.

The second stage of the analysis cycle is similar to that of equation (4.2) with the exception that the unit load analyses in these equations are replaced by the updated unit load analyses. This stage is given by:

$$\{R^r_c\} = -\{\psi^r\} \tag{4.6a}$$

$$\{\delta^r\} = \{\delta^{r-1}\} + [\delta^r_I]\{R^r_c\} \tag{4.6b}$$

$$\{\psi^r\} = \int_\Omega [B^{r-1}]^T \{\sigma^{r-1}\} \, d\Omega - \{F\} \tag{4.6c}$$

The strains and stresses are given by equations (4.3) and (4.4) respectively. Equations (4.5) and (4.6) complete one cycle of the iteration. At the next iteration these equations are used again.

4.2.1.3 The initial/tangential stiffness technique

A combination of the initial/tangential stiffness technique using the theorems of structural variation follows directly from section 4.2.1.1 and 4.2.1.2. During a load increment $[\delta^r_I]$ is kept constant for the iterations. It is only updated at the beginning of each load increment or at some suitable selected interval. Therefore within a load increment equation (4.5) is used only once to evaluate $[R^r_{cI}]$ and hence $[\delta^r_I]$. Equation (4.6) is then used for the iterations in much the same way as the initial stiffness technique.

4.2.1.4 Difficulties of the proposed techniques

With the initial stiffness technique the condensed variation factors are given directly by the residual forces at the nodes. The new displacements under the applied loading are given by direct superposition of the unit load and applied load analyses. For the tangential stiffness technique, the unit load analyses must be updated to account for the change in the stiffness under the applied loading. The reanalysis for the applied loading is undertaken using these updated unit analyses and the residual forces as the condensed variation factors.

The formulations derived previously, were early attempts to use the theorems in a nonlinear analysis technique. The basis of the ideas followed from the nonlinear analysis of a one-dimensional finite element problem given by Owen and Hinton[112]. In these problems the parameter α can be evaluated explicitly and the new element stiffness is given by:

$$[K^{*e}] = (1+\alpha^e)[K^e] \qquad (4.7)$$

which is a scalar multiple of the original stiffness. Here the only parameter that is changing is the elastic modulus and there is no difficulty in evaluating α^e (from equation (3.1)) because the [D] matrix is a scalar. However the [D] matrix for a continuum is not a scalar and there are difficulties in defining α^e for such a matrix. In elasto-plastic analysis, each coefficient of [D] is reduced by different amounts by the plastic part $[D_p]$. It is possible to define a matrix of α's as $[\alpha]$ where each coefficient is evaluated from:

$$\alpha_{ij} = \frac{(d_{ep})_{ij} - (d)_{ij}}{(d)_{ij}} \qquad (4.8)$$

where:

α_{ij} = coefficient of matrix $[\alpha]$;

$(d_{ep})_{ij}$ = coefficient of elasto-plastic matrix, $[D_{ep}]$; and

$(d)_{ij}$ = coefficient of elasticity matrix, $[D]$.

However some coefficients of [D] are zeros and hence some α_{ij}'s of equation (4.8) are undefined. This method of defining $[\alpha]$ matrix is therefore rejected.

Another possibility is to define the matrix $[\alpha]$ as:

$$[\alpha] = ([D_{ep}]-[D]).[D]^{-1} \qquad (4.9)$$

The $[D]^{-1}$ matrix is known as the compliance matrix and is defined. However the use of equation (4.9) was not investigated and could therefore be the subject of future research.

Another serious objection to the use of the theorems of structural variation for the tangential stiffness technique is the evaluation of the updated displacements in equations (4.5a) and (4.5b). This is computationally inefficient because it requires the calculation of internal forces and the matrix of condensed variation factors for each unit load analysis. The number of unit loads may be quite large and the reduction of equation (4.5a) to obtain $[R^r_{cI}]$ is prohibitively expensive.

Therefore the theorems of structural variation for the nonlinear analysis of continuum problem was abandoned in favour of the theorems of geometric variation. However it should be noted that the formulations of sections 4.2.1.1, 4.2.1.2 and 4.2.1.3 have been applied successfully to one-dimensional finite element problems[Appendix IV]. The results obtained were identical with that of the usual procedures of Newton-Raphson.

4.2.2 The theorems of geometric variation

The equilibrium equation (3.52) due to modifications are expressed in terms of the internal forces and external applied loads. This is advantageous because the residual forces are treated as applied loads using equation (3.52). Secondly there is no need to evaluate the α's because they are already included in the compensation forces. This was proved in Chapter 3 where the matrices $\lceil \alpha^e \rfloor [f^e]$ and $[C^e_{aa}]$ are shown to be equivalent. Thirdly the unit load analyses need not be updated as required by the theorems of structural variation. This means that the technique will be efficient and requires less storage.

Geometric and material nonlinear analysis may be formulated using the theorems of geometric variation by reconsidering figure 4.1(i), where the shaded region represents the assumed nonlinear elements. Unit load analyses for the nodes of the shaded elements are evaluated. These are the 'a' affected nodes, and in addition unit load analyses are required for the 'u' unaffected nodes. Unit load analyses are required for these 'u' nodes, which are nodes with applied loading that have not been included with the 'a' nodes. The unit analyses are given by:

$$[K_o][\delta_a \mid \delta_u] = [I_a \mid I_u] \tag{4.10}$$

The analysis for the applied loadings, $\{F\}$ will usually be solved in increments, $\{\Delta F\}$. The analysis for the increment may be obtained by superposition of the unit load analyses of equation (4.10).

4.2.2.1 The initial stiffness technique

In this technique the elastic stiffness, $[K_o]$ is used at all times. The matrices of compensation forces $[C_{aa}]$ and $[C_{au}]$ are therefore zero at all times since the new structural stiffness matrix, $[K_T]$ is never evaluated. Hence the equilibrium equations at the first load increment and iteration for figure 4.1(i) are:

$$\left(\left[\begin{array}{c|c} I & 0 \\ \hline 0 & I \end{array}\right] + \left[\begin{array}{c|c} 0 & 0 \\ \hline 0 & 0 \end{array}\right]\right)\left[\begin{array}{c} r_a \\ \hline r_u \end{array}\right] = \left[\begin{array}{c} \Delta F_a \\ \hline \Delta F_u \end{array}\right] \tag{4.11}$$

where:

$\{\Delta F_a\}$ = load increment at the 'a' nodes; and

$\{\Delta F_u\}$ = load increment at the 'u' nodes.

Hence $\{r_a \mid r_u\} = \{\Delta F_a \mid \Delta F_u\}$ and the total displacements are evaluated by:

$$\{\delta^r\} = \{\delta^{r-1}\} + [\delta_a \mid \delta_u]\left[\begin{array}{c} r^r_a \\ \hline r^r_u \end{array}\right] \tag{4.12a}$$

where $[\delta_a \mid \delta_u]$ is obtained from the initial analysis of equation (4.10).

However, if some elements have become nonlinear as shown in figures 4.1(ii) and (iii), a state of unbalance will exist. The residual forces are calculated for each nonlinear element using:

$$\{\psi^r\} = \int_\Omega [B^{r-1}]^T\{\sigma^{r-1}\}\, d\Omega - \{F\} \tag{4.12b}$$

where [B] and $\{\sigma\}$ must be updated to account for any nonlinearities after $\{\delta^r\}$ has been obtained. The residual forces only exist at the nodes of the nonlinear elements. At subsequent iterations:

$$\left(\left[\begin{array}{c|c} I & 0 \\ \hline 0 & I \end{array}\right] + \left[\begin{array}{c|c} 0 & 0 \\ \hline 0 & 0 \end{array}\right]\right)\left[\begin{array}{c} r^r_a \\ \hline r^r_u \end{array}\right] = \left[\begin{array}{c} \psi^r_a \\ \hline 0 \end{array}\right] \tag{4.12c}$$

which gives $\{r^r_a \mid r^r_u\} = \{\psi^r_a \mid 0\}$. This is repeated using equations (4.12a),(4.12b) and (4.12c) until $\{\psi^r\}$ becomes tolerably small within the load increment. This ensures that equilibrium is satisfied at the end of every load increment.

At the next load increment equation (4.11) is used again and the process is repeated using equation (4.12). In equation (4.11) the actual load increment is $\{\Delta F_a + \psi_a \mid \Delta F_u\}$ where the residual forces at the end of a load increment is added to the next one as mentioned in section 4.2. In the computer implementation equations (4.11) and (4.12c) are not directly solved as the scale factors are actually the load increments or residual forces. This technique is clearly a superposition of unit load analyses. It is the same as the technique described for the theorems of structural variation in section 4.2.1.1. In equations (4.11) and (4.12) the matrices and vectors involved are of the size equal to degrees of freedom of parts of the structure undergoing modifications due to nonlinearities.

4.2.2.2 The tangential stiffness technique

The tangential stiffness matrix, $[K_T]$ is calculated for every iteration of each load increment. The compensation forces for each nonlinear element may be calculated as discussed in Chapter 3. This is given using equation (3.50):

$$[C^e_{aa} \mid C^e_{au}] = ([K^e_T]-[K^e_o])[\delta^e_a \mid \delta^e_u] \tag{4.13}$$

The element nodal displacements $[\delta^e_a \mid \delta^e_u]$ are obtained from the initial analysis. Equation (4.13) would be evaluated for each nonlinear element. The matrices $[C^e_{aa}]$ and $[C^e_{au}]$ are then added to the overall compensation matrices as described in Chapter 3. At the beginning of every load increment the equilibrium equations are given by:

$$\left(\left[\begin{array}{c|c} I & 0 \\ \hline 0 & I \end{array}\right] + \left[\begin{array}{c|c} C_{aa} & C_{au} \\ \hline 0 & 0 \end{array}\right] \right) \left[\begin{array}{c} r_a \\ \hline r_u \end{array}\right] = \left[\begin{array}{c} \Delta F_a \\ \hline \Delta F_u \end{array}\right] \tag{4.14}$$

Hence the scale factors $\{r_a \mid r_u\}$ may be determined and the total displacements calculated from:

$$\{\delta^r\} = \{\delta^{r-1}\} + [\delta_a \mid \delta_u] \left[\begin{array}{c} r^r_a \\ \hline r^r_u \end{array}\right] \tag{4.15a}$$

where $[\delta_a \mid \delta_u]$ is obtained from the initial analysis.

The residual forces are evaluated as before:

$$\{\psi^r\} = \int_\Omega [B^{r-1}]^T \{\sigma^{r-1}\}\, d\Omega - \{F\} \tag{4.15b}$$

and for subsequent iterations the scale factors are:

$$\left(\left[\begin{array}{c|c} I & 0 \\ \hline 0 & I \end{array} \right] + \left[\begin{array}{c|c} C^r_{aa} & C^r_{au} \\ \hline 0 & 0 \end{array} \right] \right) \left[\begin{array}{c} r^r_a \\ \hline r^r_u \end{array} \right] = \left[\begin{array}{c} \psi^r_a \\ \hline 0 \end{array} \right] \tag{4.15c}$$

The process may then be repeated using equation (4.15) until the residual forces are zero or some small value. For the next load increment equation (4.14) is used with any residual force from the previous load increment added to $\{\Delta F\}$.

The size of matrices $[C_{aa}]$ and $[C_{au}]$ depend on how many elements have become nonlinear. They become progressively larger as the load increases and more elements become nonlinear. These matrices are also full and unsymmetric and therefore the solution for $\{r^r_a\}$ will be expensive. This is especially true when the number of nonlinear elements becomes large compared to the total number of elements in the structure. Obviously the technique is particularly efficient for localised nonlinear behaviour.

4.2.2.3 The initial/tangential stiffness technique

A combination of the initial and tangential stiffness technique using the theorems of geometric variation is easily implemented. At the beginning of each load increment equation (4.14) is used and the matrices $[C_{aa}]$ and $[C_{au}]$ are assembled from each nonlinear element contributions. For subsequent iterations equation (4.15c) is used but the $[C_{aa}]$ and $[C_{au}]$ matrices are those calculated at the beginning of each load increment. These matrices are kept constant during the load increment. As suggested in Chapter 2 other suitable intervals are also possible for updating the matrices $[C_{aa}]$ and $[C_{au}]$.

In the actual computer implementation, only the reduced form of equation (4.15c) and the Gaussian reduction factors are stored. The scale factors are then obtained by reduction of the right side of (4.15c) only and then back-substituted.

4.2.2.4 Variant forms of the Newton-Raphson methods

When the applied loads only act at a few unaffected nodes, 'u' the total number of unit load analyses required is small. However when body forces are involved, which implies that every node of the finite element structure is loaded, unit load analyses at every node is required.

To avoid this, a modified form of the initial and tangential stiffness technique may be formulated. For the initial analysis the unit loads at the 'a' nodes are performed. Instead of analysing unit loads at the 'u' nodes, the analysis is for the total applied loads $\{F\}$. The total number of load analyses required is $(a+1)$ instead of $(a+u)$. This analysis is given by equation (4.1) from the theorems of structural variation.

The analysis of the modified structure may be undertaken as follows; at the start of each load increment, the incremental displacements are evaluated by:

$$\{\Delta\delta\} = \lambda\{\delta\} \tag{4.16}$$

where:

λ = load factor which is related to the load increment, using the proportional loading relationship $\lambda\{F\} \equiv \{\Delta F\}$; and

$\{\delta\}$ = displacements due to applied loads.

For the subsequent iterations equation (4.12c) is used for the variant initial stiffness technique. It gives identical results to the technique of section 4.2.2.1.

The results of the tangential stiffness technique is different to the variant form using equation (4.15c) for the subsequent iterations. This is illustrated for a one degree of freedom problem in figure 4.2.

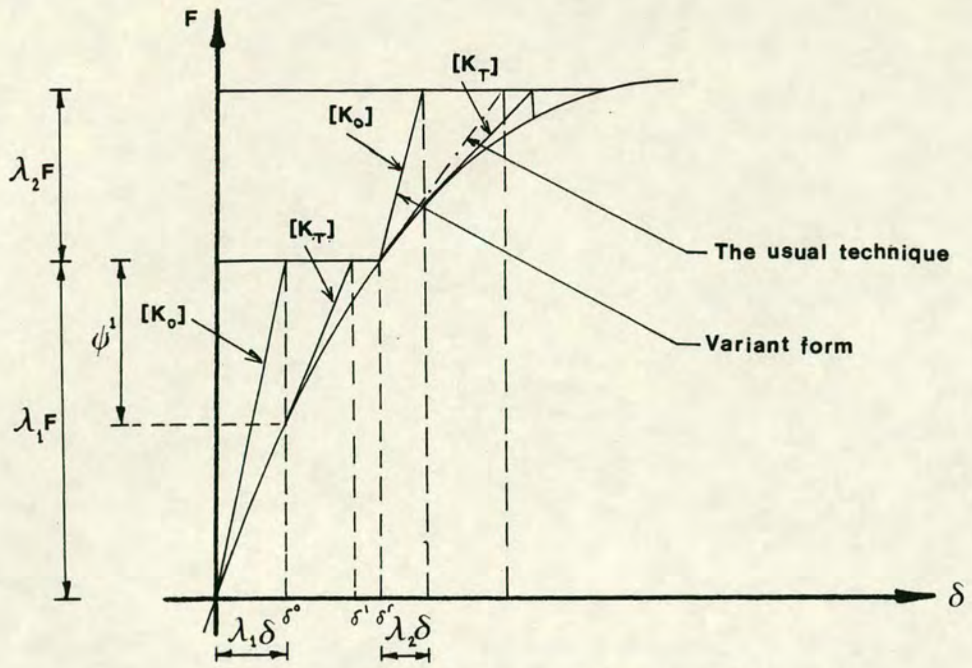


Figure 4.2 Tangential stiffness technique

At the first load increment both the variant form and the tangential stiffness technique give the same results. This is because the same elastic stiffness is used at the first iteration of the first load increment. However at the second load increment, equation (4.16) is used to evaluate the incremental displacements using the elastic stiffness matrix of the structure, $[K_0]$ instead of the tangential stiffness $[K_T]$. Hence at the first iteration the incremental displacements are underestimated. After this the tangential stiffness is used. It converges at a slightly slower rate than that of the usual procedure. Although the convergence rate of the variant technique may be slower than that of the tangential stiffness technique, the incremental displacements for the first iteration of every load increment are obtained at less computational expense. The variant technique is only applicable to proportional loading.

4.3 Summary

The theorems of structural and geometric variation have been presented as nonlinear solution algorithms. The nonlinear solution algorithms are formulated in terms of the well known and tested techniques of Newton-Raphson and its degenerate forms. Within each load increment, iterations are carried out to ensure that equilibrium is satisfied at the end of each load increment.

The theorems of structural variation are shown to be inadequate as a nonlinear solution technique. This is because the modified stiffness is obtained by a scalar multiple of the original stiffness. In general for nonlinear finite element problems the modified stiffness cannot be obtained in this way. The technique is also inefficient because it requires the updating of the unit load analyses for the tangential stiffness form.

The theorems of geometric variation are a more general formulation. They may easily be applied as a nonlinear solution technique and are particularly efficient for localised nonlinearity. The efficiency of the theorems of geometric variation for nonlinear analysis are further investigated in Chapters 7,8 and 9.

Another important point that should be noted is that the proposed techniques may easily be implemented into existing computer codes. Therefore the existing codes require little modification and this aspect is discussed further in **Appendix VI**.

CHAPTER 5

EFFICIENCY OF THE THEOREMS OF STRUCTURAL VARIATION

AND SOME POTENTIAL APPLICATIONS

5.1 Introduction

In this chapter the efficiency of the theorems of structural variation as a reanalysis technique is investigated. The efficiency tests were carried out on some simple finite element structures where the number of modified elements were progressively increased. The reanalysis was compared with the time taken for a complete or fresh analysis. The measure of efficiency was based on the central processor unit (or CPU) time taken for the analysis on the ICL 2900 mainframe computer. The CPU time will depend on the type of computer installation, programming techniques and the compilers used. This is briefly discussed in **Appendix VI** on computer implementation.

The elements used for the efficiency tests are the constant strain triangular, natural stiffness triangular and the isoparametric quadrilateral elements. A simple structure idealised using these elements is considered for the tests. Finally two examples are given on the use of these theorems. The first example is the repeated analysis of a shear wall and the other involves soil excavation at various stages of the construction.

5.2 Investigation of the efficiency

The general matrix formulations have been presented in Chapter 3. For efficiency the equilibrium equations of the modified parts of the structure are formed by summation of the out of balance forces for each modified element.

The first tests involve the constant strain triangular (CST) and natural stiffness triangular (NST) elements. The tests are to compare the evaluation of the modified structure using the variation factors and condensed variation factors. The relevant equations are respectively (3.3) and (3.10) of Chapter 3. These tests are to confirm that the use of condensed variation factors is more efficient than the uncondensed variation factors.

The second tests are a comparison of the efficiency of the

CST and NST elements. The NST elements require only three edge forces to define the state of stress within the element[Appendix III]. For one NST element to be modified only three equilibrium equations are required as compared to six for the CST element. Therefore the NST element should be more efficient in comparison to the CST element. It was first suggested by Topping[138] that the NST element should be used for the reanalysis as it is more efficient. However it should be noted that the initial analysis must be undertaken using the CST element with three pairs of unit loads applied along the edges of the element. The 4-node isoparametric quadrilateral element (QUAD4) was also tested for efficiency. It was used as a comparison with the triangular elements. All the tests mentioned so far are relative tests for comparison of the efficiency of the formulations and the elements used in the reanalysis. The absolute test is ultimately based on a comparison with a fresh analysis.

5.2.1 The structures chosen as benchmark tests

The CST and NST elements were tested for the structures shown in figures 5.1(i),(ii) and (iii). For the QUAD4 element the structures of figures 5.2(i),(ii) and (iii) were tested where two triangular elements were taken as equivalent to one quadrilateral element. The number of elements, nodes and semi-bandwidth of each structure are tabulated in table 5.1.

Structure (Figure)	Type of element used	No. of elements	No. of nodes	Semi-bandwidth
5.1(i)	CST and NST	50	36	16
5.1(ii)	CST and NST	200	121	26
5.1(iii)	CST and NST	800	441	46
5.2(i)	QUAD4	25	36	16
5.2(ii)	QUAD4	100	121	26
5.2(iii)	QUAD4	400	441	46

Table 5.1 Structures tested

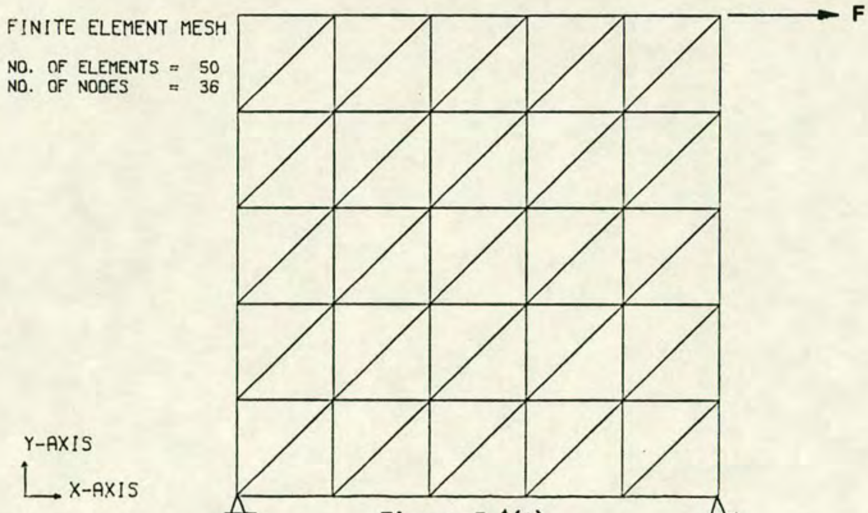


Figure 5.1(i)

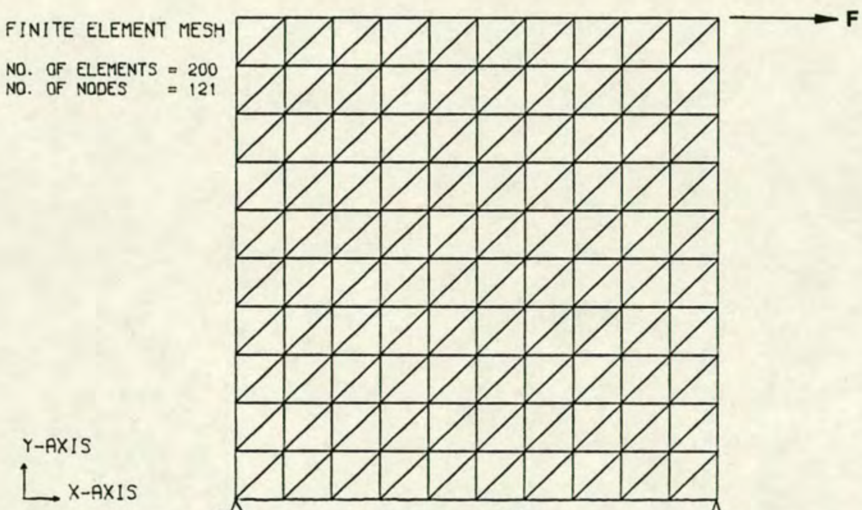


Figure 5.1(ii)

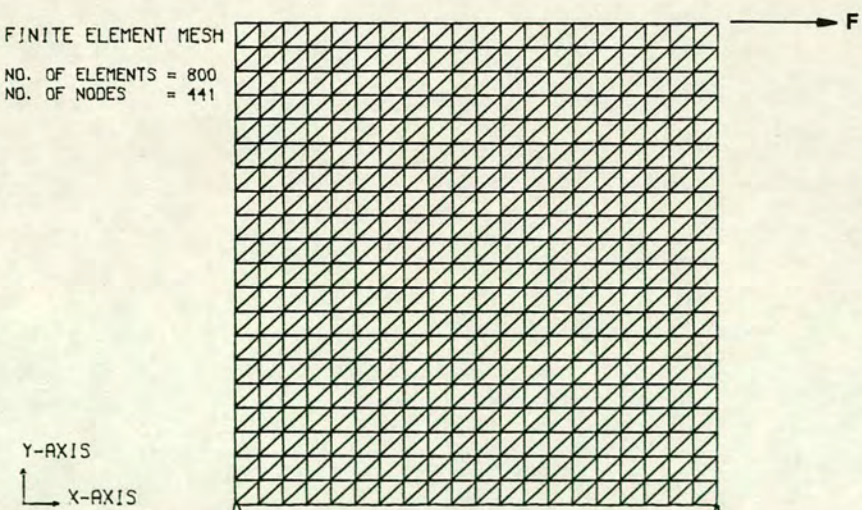


Figure 5.1(iii)

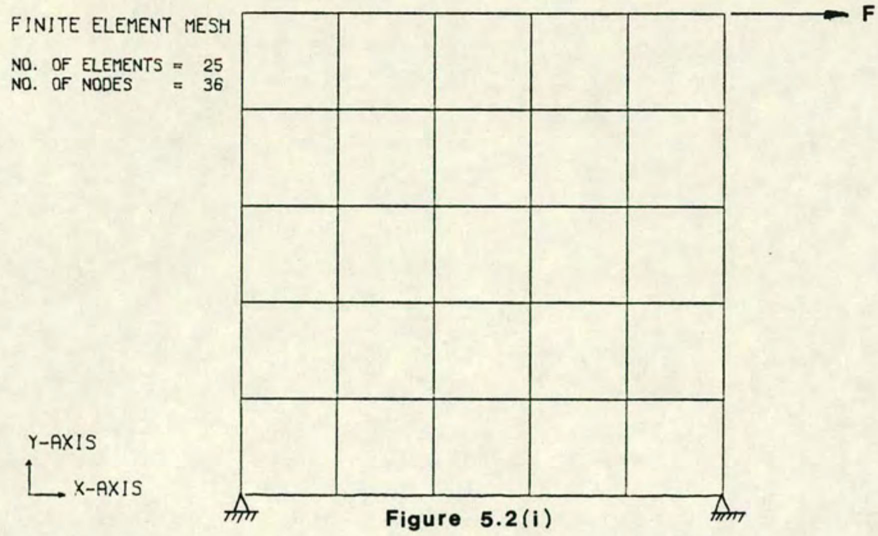


Figure 5.2(i)

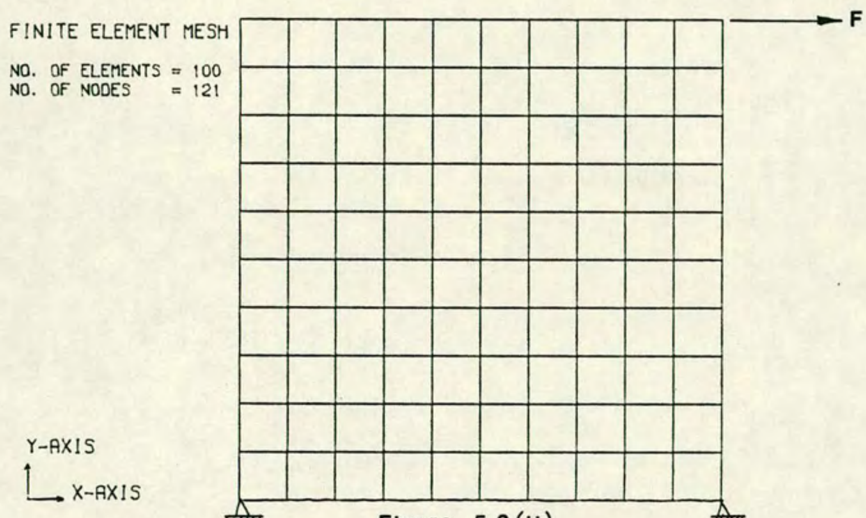


Figure 5.2(ii)

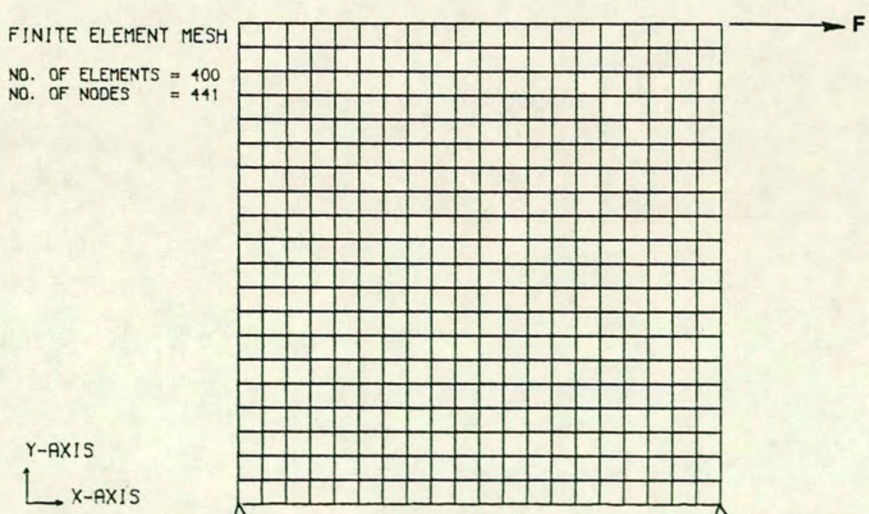


Figure 5.2(iii)

The elements to be modified are adjacent to one another at the left hand corner of each structure. The number of modified elements are increased one at a time. Figure 3.1 of Chapter 3 is a typical example. The CPU time taken for a reanalysis when element 1 is modified is measured, then for elements 1 and 4, elements 1,4 and 2, elements 1,4,2 and 5, and so on. As these structures are simply tested for efficiency of the formulations any suitable set of units for the dimensions and material properties may be used.

5.2.2. The CPU time taken for the number of elements modified

The graphs of CPU time taken for reanalysis against the number of elements modified are shown in figures 5.3(i),(ii) and (iii). In the graphs the label CST(3.3) for example indicates that CST elements were used for the reanalysis with equation (3.3) to evaluate the variation factors. The type of formulations used for each structure are tabulated in table 5.2.

Structure (Figure)	Type of element used	Formulation used for variation factors
5.1(i),(ii),(iii)	CST	Equation (3.3) Variation factors
5.1(i),(ii),(iii)	NST	Equation (3.3) Variation factors
5.1(i),(ii),(iii)	CST	Equation (3.10) Condensed variation factors
5.1(i),(ii),(iii)	NST	Equation (3.10) Condensed variation factors
5.2(i),(ii),(iii)	QUAD4	Equation (3.10) Condensed variation factors

Table 5.2 Formulations used in the reanalysis

Note that equations (3.3) and (3.10) are used to evaluate the variation factors, $\{R\}$ and condensed variation factors, $\{R_c\}$ respectively. The matrix forms of these equations were used in the computer implementation.

The graphs show the relative efficiency of the types of formulation and elements used. For the absolute efficiency a comparison is made with a complete or fresh analysis. The fresh

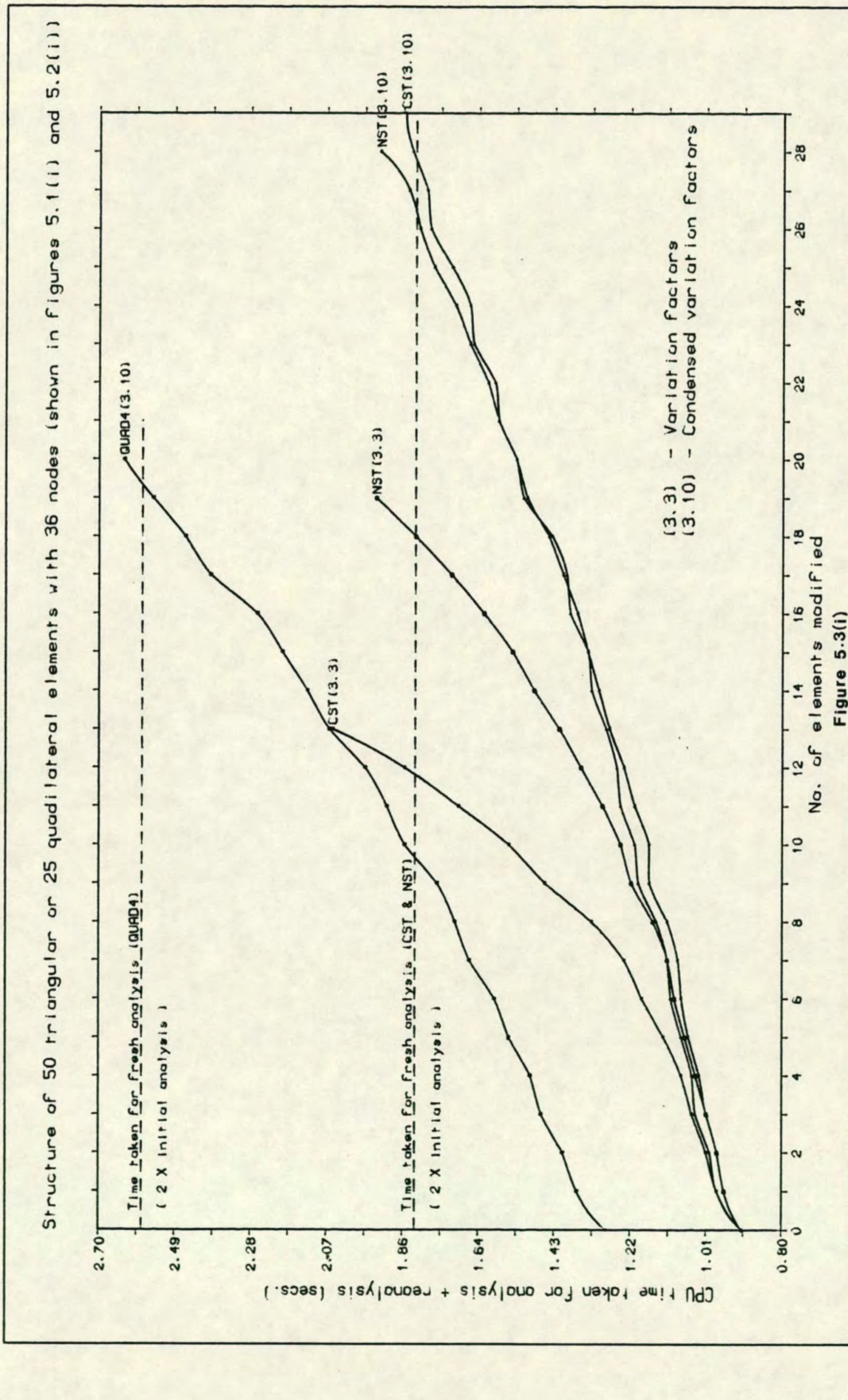

Structure of 50 triangular or 25 quadrilateral elements with 36 nodes (shown in figures 5.1(i) and 5.2(ii))
Time taken for fresh analysis (QUAD4)
(2 X initial analysis)
Time taken for fresh analysis (CST & NST)
(2 X initial analysis)
QUAD4(3.10)
CST(3.3)
NST(3.3)
NST(3.10)
CST(3.10)
(3.3) - Variation factors
(3.10) - Condensed variation factors
CPU time taken for analysis + reanalysis (secs.)
No. of elements modified
2.70
2.49
2.28
2.07
1.86
1.64
1.43
1.22
1.01
0.80
0
2
4
6
8
10
12
14
16
18
20
22
24
26
28

Figure 5.3(i)

Structure of 200 triangular or 100 quadilateral elements with 121 nodes (shown in figures 5.1(ii) and 5.2(ii))

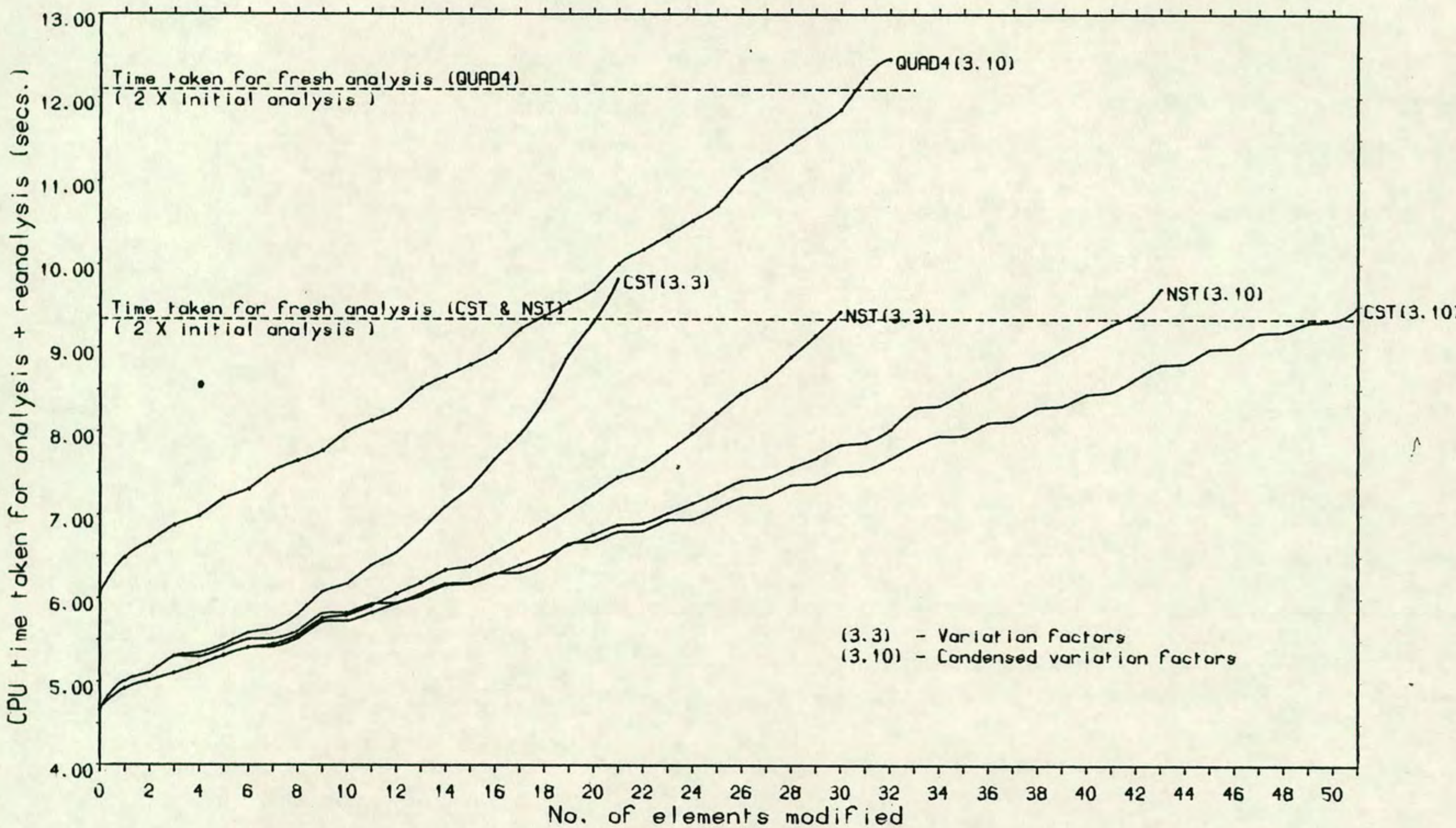


Figure 5.3(ii)

Structure of 800 triangular or 400 quadilateral elements with 441 nodes (shown in figures 5.1(iii) and 5.2(iii))

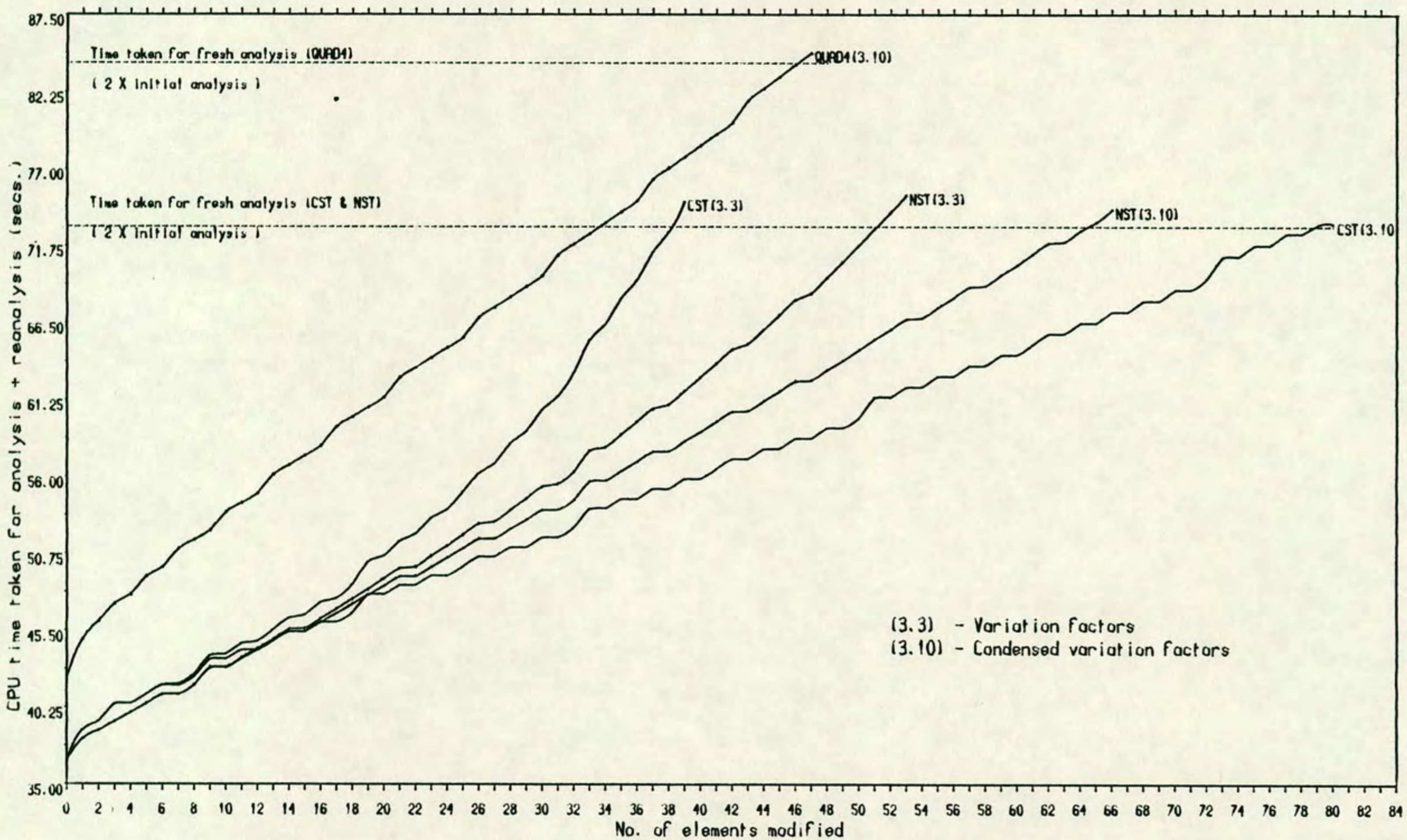


Figure 5.3(iii)

analysis is taken as twice the time taken for a single analysis. This is shown in the graphs by the horizontal 'broken' lines. The time for a single analysis is the time taken when the number of elements modified is zero. In a design problem a single analysis is often inadequate because some parts of the structure may exceed the design constraints. Therefore further analyses are required with modifications to satisfy the design constraints.

5.2.3 Maximum percentage of elements modified efficiently

Table 5.3 shows the maximum percentage of the total number of elements that may be modified without the reanalysis technique becoming inefficient. This provides a measure of the efficiency of the type and number of elements used in an idealisation that will require reanalysis. The maximum percentage of elements that is modified efficiently is define as:

$$\text{Max. \% of elements modified efficiently} = \frac{\text{Max. no. of elements modified efficiently}}{\text{Total no. of elements}} \times 100\,\% \quad (5.1)$$

	Structural idealisation (Max. % of elements modified efficiently)		
Type of elements and formulation	Figure 5.1(i) (50 elements)	Figure 5.1(ii) (200 elements)	Figure 5.1(iii) (800 elements)
CST(3.3)	22.0%	10.0%	4.8%
CST(3.10)	54.0%	25.0%	9.8%
NST(3.3)	36.0%	14.5%	6.4%
NST(3.10)	52.0%	20.5%	8.0%
	Figure 5.2(i) (25 elements)	Figure 5.2(ii) (100 elements)	Figure 5.2(iii) (400 elements)
QUAD4(3.10)	76.0%	30.0%	11.5%

Table 5.3 Maximum percentage of elements modified efficiently

5.2.4 Discussion of results

As expected the formulation using equation (3.10), where the condensed variation factors were evaluated is more efficient than using equation (3.3). This may be clearly observed from the graphs of figures 5.3(i),(ii),(iii) and the table 5.3. In Chapter 3, it was noted that equation (3.10) involves a smaller set of equilibrium equations than equation (3.3). The importance of this is clearly demonstrated by the graphs. As the number of modified elements increases the use of equation (3.3) becomes increasingly inefficient.

The CST element is more efficient than the NST element when the number of modified elements is large and if condensed variation factors are used. This is somewhat surprising at first but may be explained as follows. If one NST element is used the number of unit load analyses required is three. These unit loads are applied along the edges of the element. The number of simultaneous equations to be solved is also three. In contrast, the CST element requires six unit loads and equilibrium equations. Hence the NST element would appear to be more efficient. This is true if the number of modified elements are small or the elements are not adjacent to one another. However when the number of elements to be modified is large and the elements are adjacent to one another is to be considered. For example if all the elements of the idealisation shown in figure 3.1 are to be modified, the required number of unit load analyses are:

For CST elements, the no. of unit loads = 32

For NST elements, the no. of unit loads = 33

The number of equilibrium equations will hence be the same in each case. Since most of the CPU time is taken by the reduction of the unit loads and equation (3.10), the CST elements are more efficient in this case.

For the case when the modified elements are not adjacent, the modification of elements 1,4,9 and 12 of figure 3.1 will be considered. The number of unit load analyses will be:

For CST elements, the no. of unit loads = 16

For NST elements, the no. of unit loads = 10

Hence the NST element is very efficient in this case. In the structures shown in figures 5.1 and 5.2 the modified elements are adjacent to one another. Therefore the graphs in figures 5.3(i),(ii),(iii) and the figures of table 5.3 indicate that the CST elements become more efficient as the number of modified elements increases.

The figures in table 5.3 indicates that as the structure increases in size, the maximum percentage number of elements that may be modified using the reanalysis decreases. Therefore reanalysis of a smaller structure is generally more efficient. However a small structure of 50 CST or 25 QUAD4 elements will frequently not be a realistic finite element idealisation. In reference [120] some 600 CST elements were needed to model a simply supported beam accurately such that the results were near the theoretical values. Therefore the structures of figures 5.1(iii) and 5.2 (iii) would appear more realistic. Although the percentage of modified elements are smaller for these structures, usually only a few elements would require modification in critically over stressed areas.

The use of QUAD4 elements is more efficient than the CST elements. It was assumed that two CST elements are equivalent to one QUAD4 element and hence the formation of equation (3.10) takes less time. This is a reasonable assumption because the QUAD4 element gives a better prediction of the stresses. The use of elements with higher order displacement functions will require fewer elements for the idealisation to yield an analysis of a given accuracy. In turn an analysis of fewer higher order elements may be reanalysed more efficiently.

It is fairly obvious from the matrix formulations of Chapter 3 that most of the CPU time is in:

1. Reduction of the overall stiffness matrix and the multiple load cases of equation (3.7). As the number of modified elements increases the number of unit loads also

increases. The proportion of the total time taken to reduce the loads is much smaller than the solution of equation (3.3) or (3.10) for a large number of elements. It is only significant when the number of elements is small.

2. The solution for the variation or condensed variation factors from equations (3.3) and (3.10) respectively. This becomes a problem with equation (3.3) as the number of simultaneous equations increases rapidly as more elements are modified. Both equations result in a full and unsymmetric matrix and no economies can be made here in either the solution technique or computer storage. This is the most inefficient part of this reanalysis technique and it takes most of the time. Partial pivoting in reducing these equations is not required since the solutions do not suffer from any round-off errors.

The size of the semi-bandwidth of the structure stiffness matrix does not affect the reanalysis. The number of equilibrium equations solved during the reanalysis only depends on the number of degrees of freedom of the modified elements. An idealisation with a large semi-bandwidth may be more efficiently reanalysed using the technique. This suggests that the technique will be particularly useful in the reanalysis of large structures with large semi-bandwidths requiring few elements to be modified.

5.3 Some applications

The potential uses of the theorems of structural variation are investigated by considering two examples. The first involves the successive analysis of a shear wall with various cut-outs during a design procedure. The second is concerned with the analysis of an excavation at various stages during removal of soil.

In each example the efficiency of the theorems of structural variation for reanalysis is assessed by comparison with fresh analysis normally used for analysis and design. In this way the efficiency of the use of the theorems to evaluate many alternative designs may be evaluated.

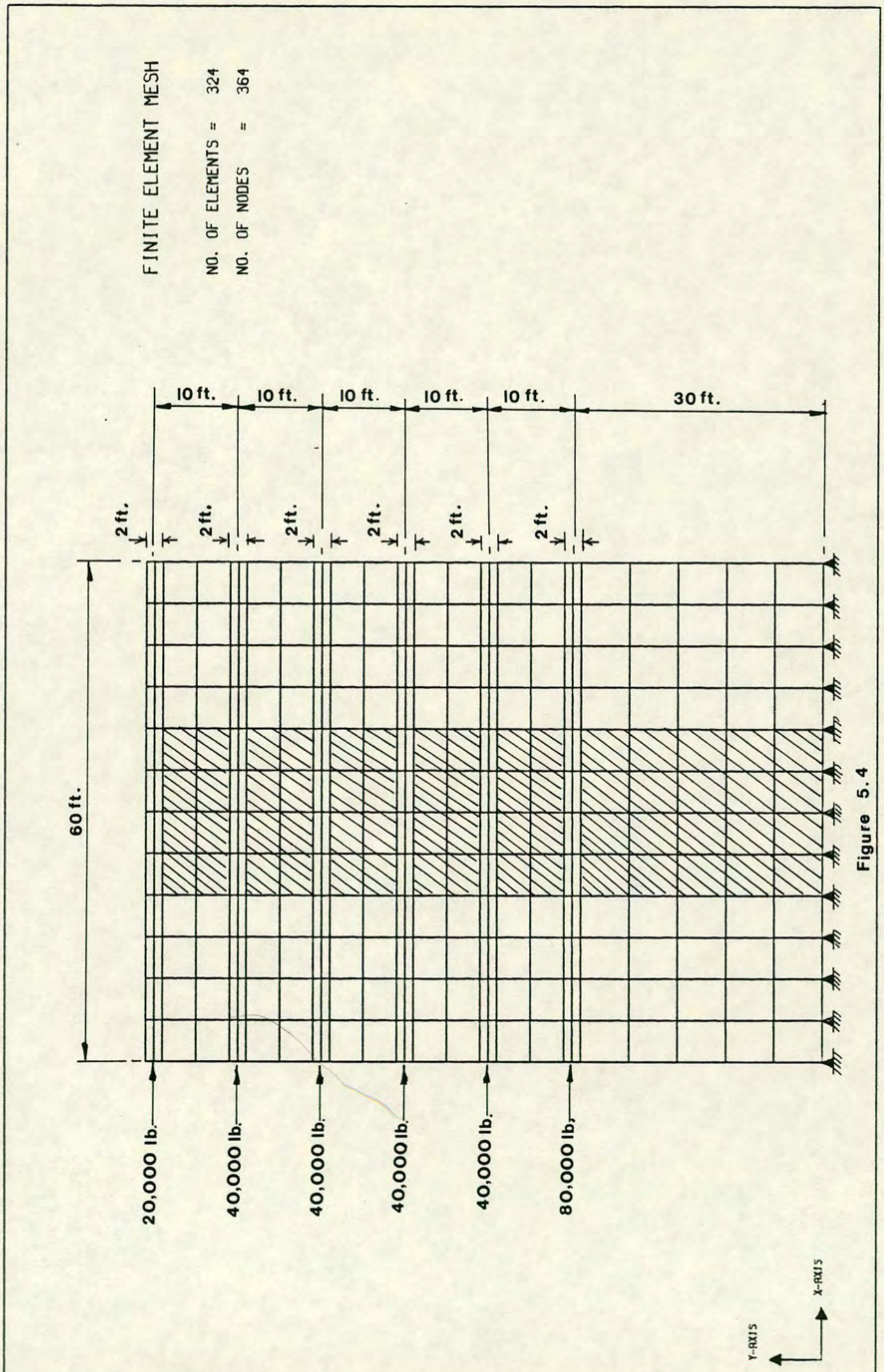
5.3.1 Shear wall analysis

The analysis of shear walls by conventional means involves many approximations because the problem is highly redundant. Ideally, the shear wall should be treated as a continuum problem, but a closed-form solution is rarely available in shear walls with complex patterns of cut-outs. The finite element method may be used to solve this problem, where the shear wall is treated as a plane stress elasticity problem. A typical shear wall analysed by Girijavallabhan[47] is used as an example.

5.3.1.1 Initial analysis

The dimensions of the original structure to be analysed is as shown in figure 5.4. QUAD4 elements were used to model the shear wall with smaller elements for the lintel beams. The bottom boundaries are assumed fixed. The lateral loads, due to the wind, are transmitted to the shear walls by horizontal cross beams. The elastic modulus and Poisson's ratio of the structure were taken as equal to 4×10^6 p.s.i. and 0.4 respectively. The thickness of the wall was assumed to be equal to 1.0 ft. (a conversion table for S.I. units is provided in Appendix V).

The original structure is different from that of reference [47] in that all the openings were filled in. The elements to be modified are shown hatched in figure 5.4 and removal of these elements gives the structure of reference [47]. The intermediate structures were obtained by the gradual removal of elements and this may easily be undertaken using the theorems. The structure was initially analysed considering applied loading plus unit loads at each of the nodes of the hatched elements.



FINITE ELEMENT MESH
NO. OF ELEMENTS = 324
NO. OF NODES = 364
10 ft.
10 ft.
10 ft.
10 ft.
10 ft.
30 ft.
2 ft.
2 ft.
2 ft.
2 ft.
2 ft.
2 ft.
60 ft.
20,000 lb.
40,000 lb.
40,000 lb.
40,000 lb.
40,000 lb.
80,000 lb.
Y-AXIS
X-AXIS

Figure 5.4

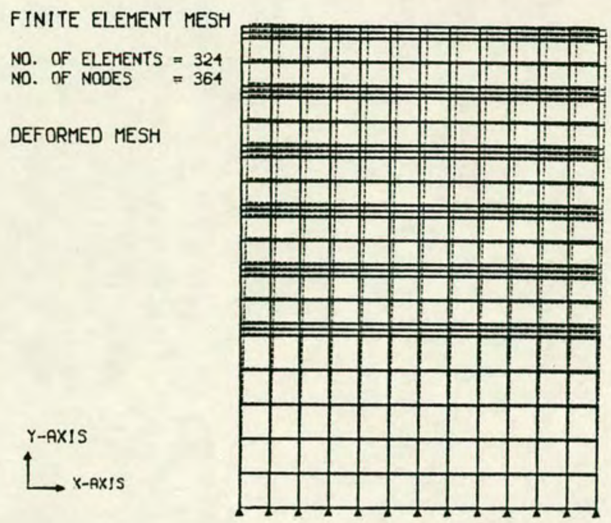


Figure 5.5(i)

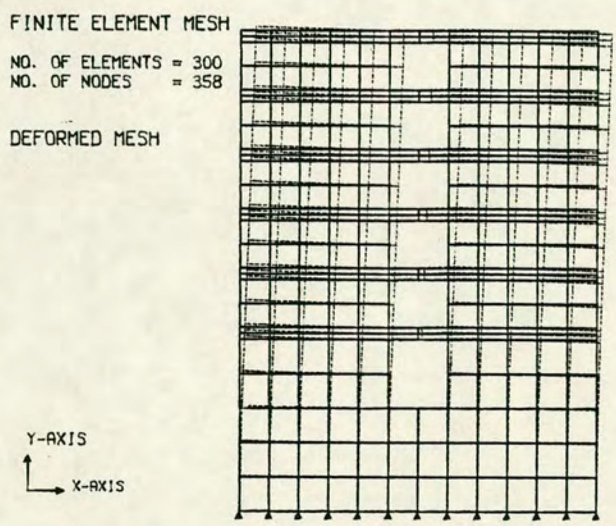


Figure 5.5(ii)

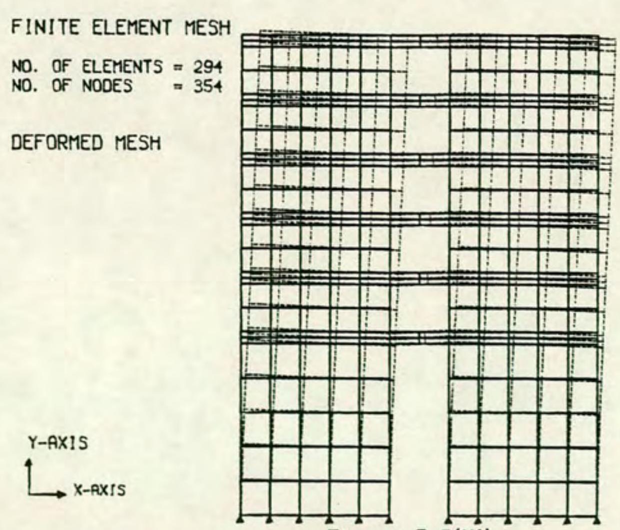


Figure 5.5(iii)

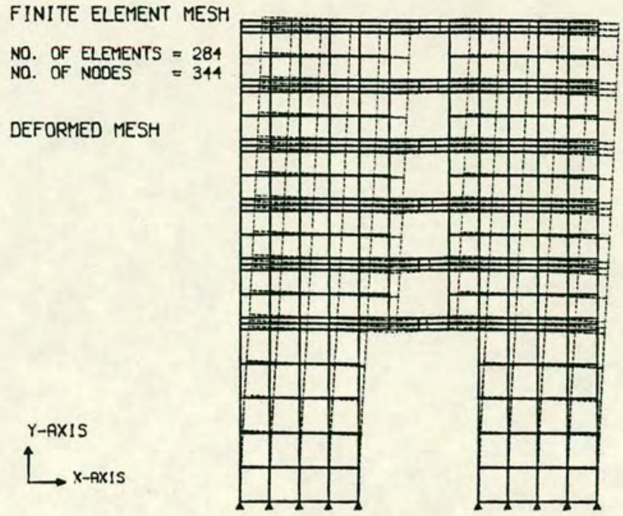


Figure 5.5(iv)

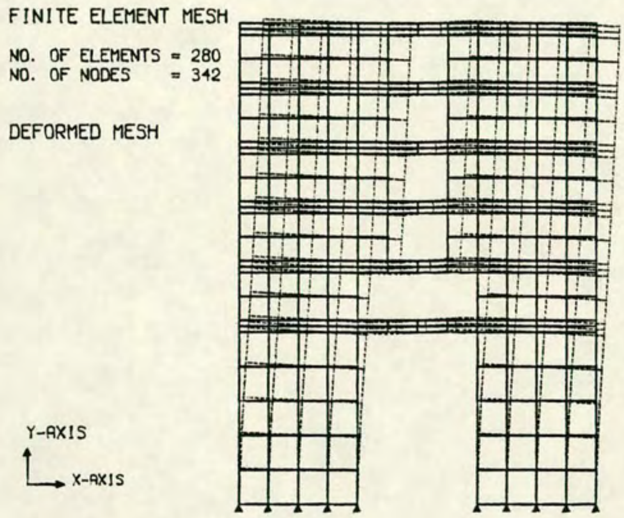


Figure 5.5(v)

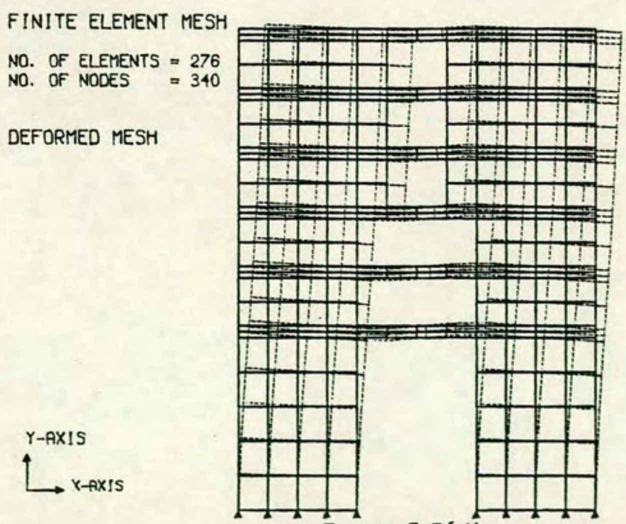


Figure 5.5(vi)

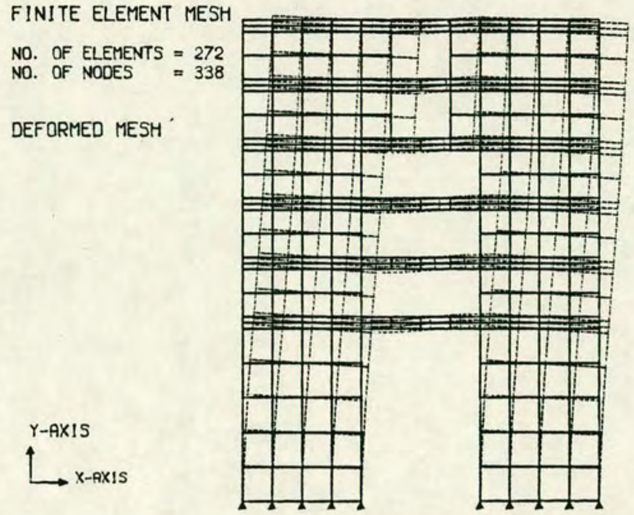


Figure 5.5 (vii)

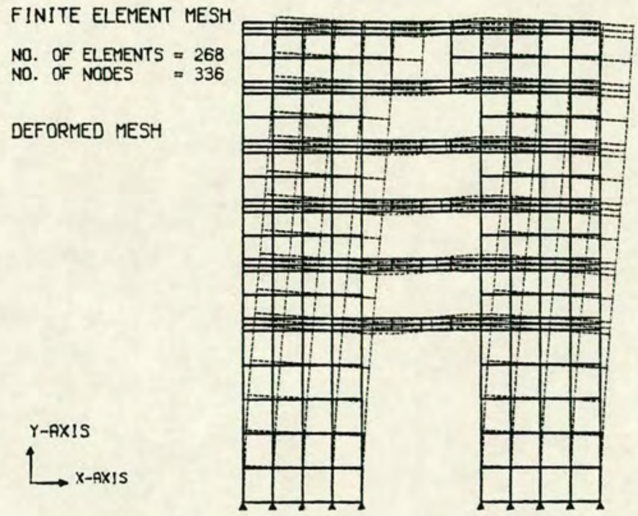


Figure 5.5 (viii)

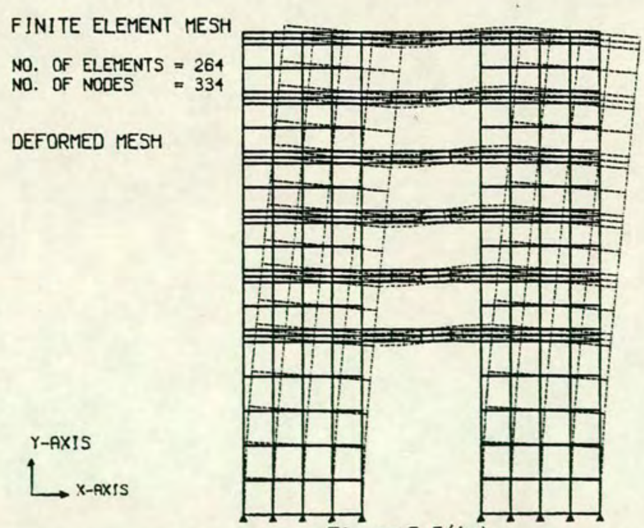


Figure 5.5 (ix)

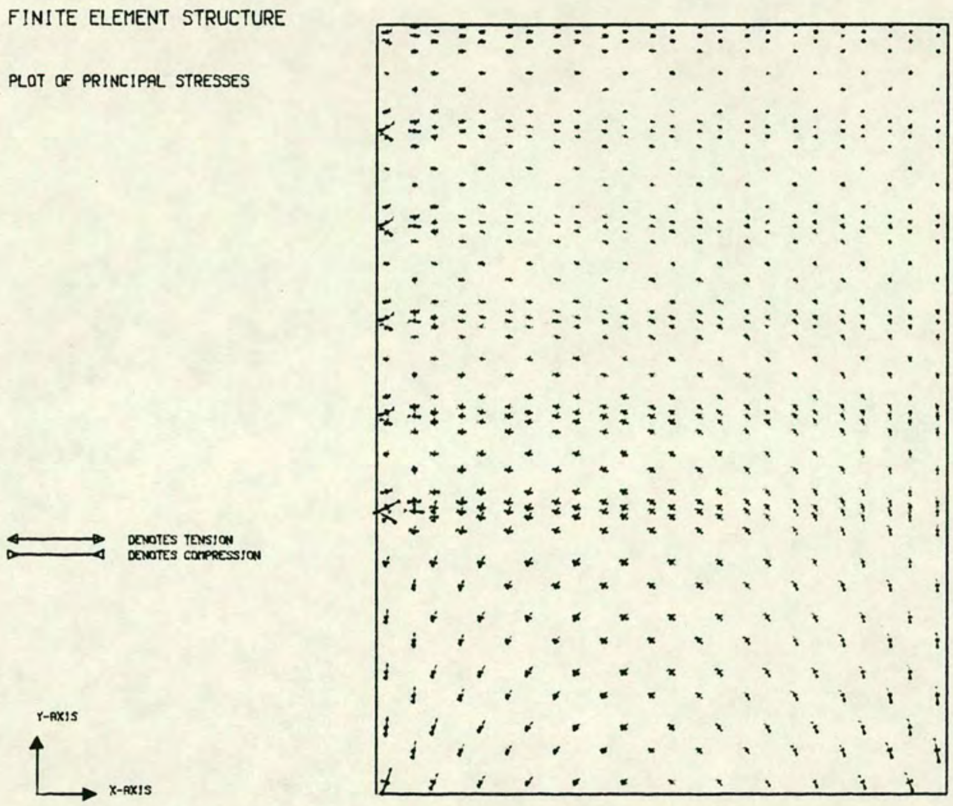


Figure 5.6(i)

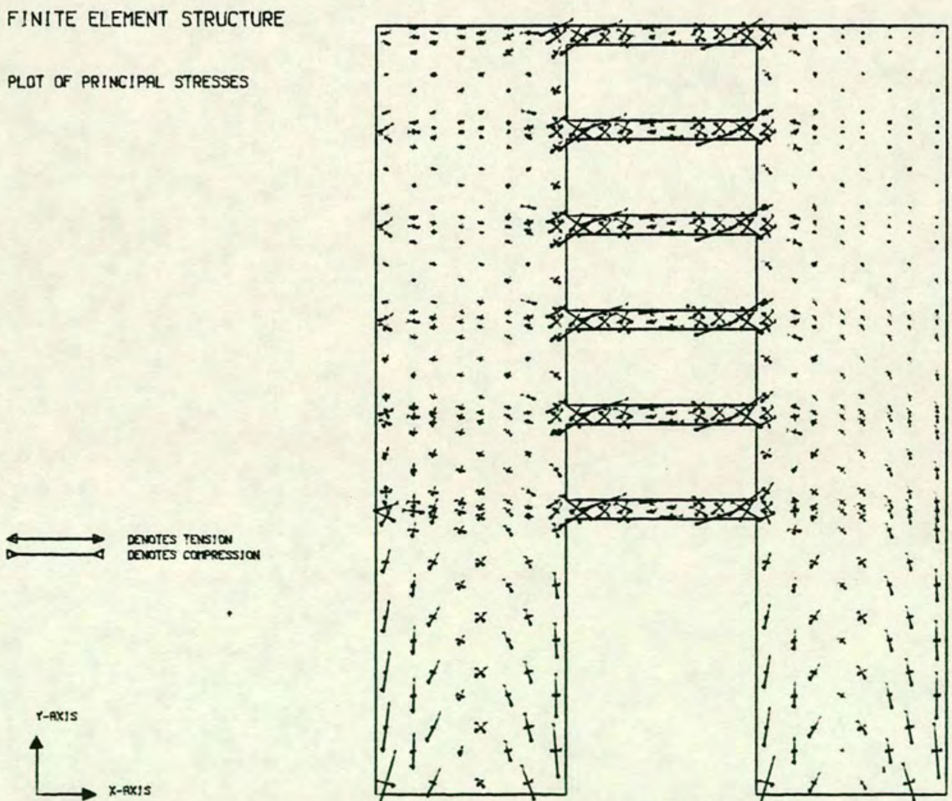


Figure 5.6(ii)

5.3.1.2 Analysis of modified structures

Openings within the shear wall (for windows or doors) may be simulated by the removal of elements. This means specifying an elastic modulus of $E'_e=0$ in equation (3.1) for the elements to be removed. A series of modified structures were analysed using the theorems as shown in figures 5.5(i) to (ix). The final structure is the same one as analysed by Girijavallabhan[47].

The stresses and displacements are 'exact' for each analysis. Plots of the deformed mesh are superimposed on the original mesh as illustrated in figures 5.5. The element principal stresses for the original and final structure are plotted in figure 5.6. Large changes in stresses occurred. The CPU time taken for each reanalysis are tabulated in table 5.4.

	CPU time taken (seconds)	
Structure (Figure)	Fresh analysis	Reanalysis by the theorems
5.5(i)	30.38	74.72 (initial analysis)
5.5(ii)	30.38	7.82
5.5(iii)	30.38	9.41
5.5(iv)	30.38	12.29
5.5(v)	30.38	14.00
5.5(vi)	30.38	15.95
5.5(vii)	30.38	18.14
5.5(viii)	30.38	20.59
5.5(ix)	30.38	23.32
Total CPU time Σ	273.42	196.24

Table 5.4 CPU time for reanalysis

The CPU time for a fresh analysis by the usual procedure requires 30.38 seconds. The analysis of the series of modified structures would require

(30.38 X 9) = 273.42 seconds. This assumes a completely fresh analysis for each modified structure.

In table 5.4 the time taken for the initial analysis of the structure in figure 5.5(i) is large compared to an analysis for the applied loading because the number of unit loads to be considered is large. For each reanalysis of the modified structure the time taken is only a fraction of the initial or fresh analysis. Therefore the total time taken for the series of reanalysis is much smaller than that for repeated fresh analysis of the modified structures. Each reanalysis is referred to the original structural analysis. The theorems of structural variation are efficient in such situations where the number of modified structures to be analysed increases. This is advantageous when a number of design alternatives must be considered for architectural or environmental reasons or to accommodate irregular openings.

In this particular problem, no attempt has been made in using any of the optimization techniques available. This may be easily incorporated in the analysis where the displacements and stresses may be considered as constraints when elements are being removed. Other uses of the theorems would be in computer-aided-design where reanalysis of modified structures may be obtain efficiently and an interactive mode of design would then become attractive.

5.3.2 Soil excavation

The finite element method may be applied to problems involving the simulation of excavation. The analysis is designed so that the actual construction sequence may be modelled as efficiently as possible. The usual technique of analysis is to apply an increment of load for each stage of the excavation. The loads are applied to the excavated boundary so as to reduce the stresses on the boundary to zero. Elements in the excavated area have their stiffnesses reduced to near zero or are removed altogether.

The usual procedure of analysis is outlined as follows. Consider figure 5.7(i) where the soil is in a state of initial stress, $\{\sigma_0\}$ given by:

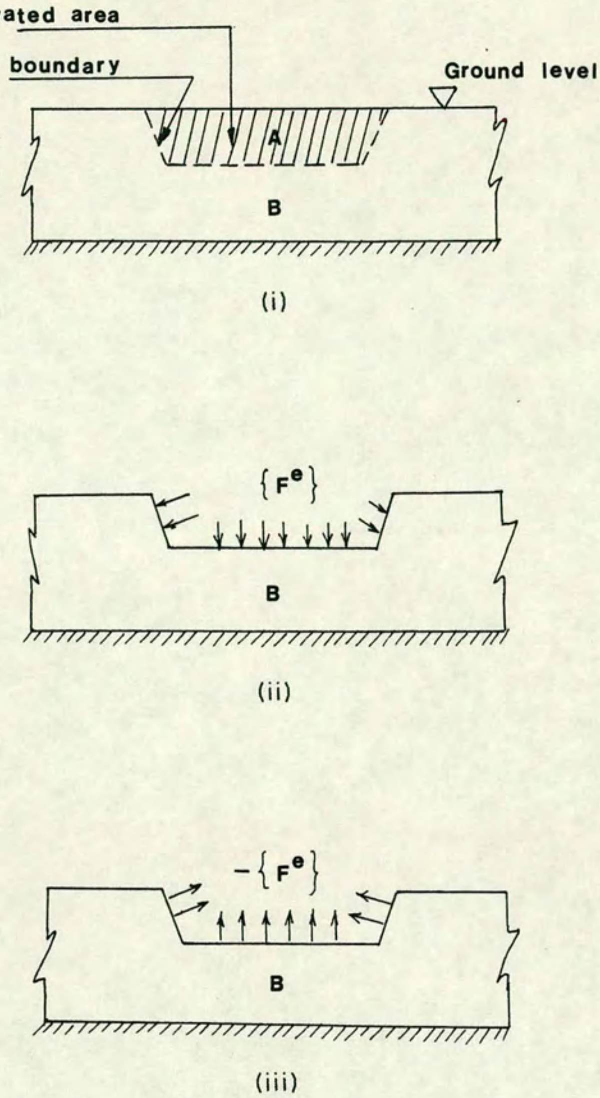


Figure 5.7 Stages of soil excavation

$$\sigma_{xo} = k\gamma h \tag{5.2a}$$

$$\sigma_{yo} = \gamma h \tag{5.2b}$$

$$\tau_{xyo} = 0 \tag{5.2c}$$

where:

k = coefficient of earth pressure at rest;

γ = unit weight of soil; and

h = depth below ground surface.

The stresses transmitted from the portion A to B are first calculated using equation (5.2). These stresses are converted to equivalent nodal forces at the nodes of the new boundaries of the elements in B. The element nodal forces, $\{F^e\}$ are evaluated from:

$$\{F^e\} = \int_{\Omega e} [B]^T \{\sigma_o\} \, d\Omega_e \tag{5.3}$$

The stiffness of portion A is removed, leaving the stiffness of B and the nodes and elements of B. This is shown in figure 5.7(ii). The nodal forces, $\{F^e\}$ calculated previously are now applied to B; they are equal in magnitude but opposite in sign as in figure 5.7(iii). The incremental displacements and stresses due to these loads are then added to the original values for the portion B. For the next stage of the excavation the new stresses are now treated as initial stresses and the process is repeated. In general for N stages of the excavation:

$$\{\delta_i\} = \{\delta_o\} + \sum_{i=1}^{N} \{\Delta\delta_i\} \tag{5.4a}$$

$$\{\sigma_i\} = \{\sigma_o\} + \sum_{i=1}^{N} \{\Delta\sigma_i\} \tag{5.4b}$$

where:

$\{\delta_o\}$ = initial displacements;

$\{\Delta\delta_i\}$ = incremental displacements at i'th stage; and

$\{\Delta\sigma_i\}$ = incremental stresses at i'th stage.

The applied loads change at each stage of the excavation as compared to the previous example where it remains the same throughout the analysis. As will be shown later, the theorems may easily accommodate this.

5.3.2.1 Initial analysis

The dimensions of the problem is shown in figure 5.8 and is the same problem analysed by Chandrasekaran and King[27]. The bottom boundary is fixed and the side boundaries may displace in the y-direction only. The soil is anqlysed as a plain strain problem and assumed homogeneous and elastic. The elastic modulus, unit weight, Poisson's ratio and coefficient of earth pressure at rest are

FINITE ELEMENT MESH

NO. OF ELEMENTS = 210
NO. OF NODES = 240

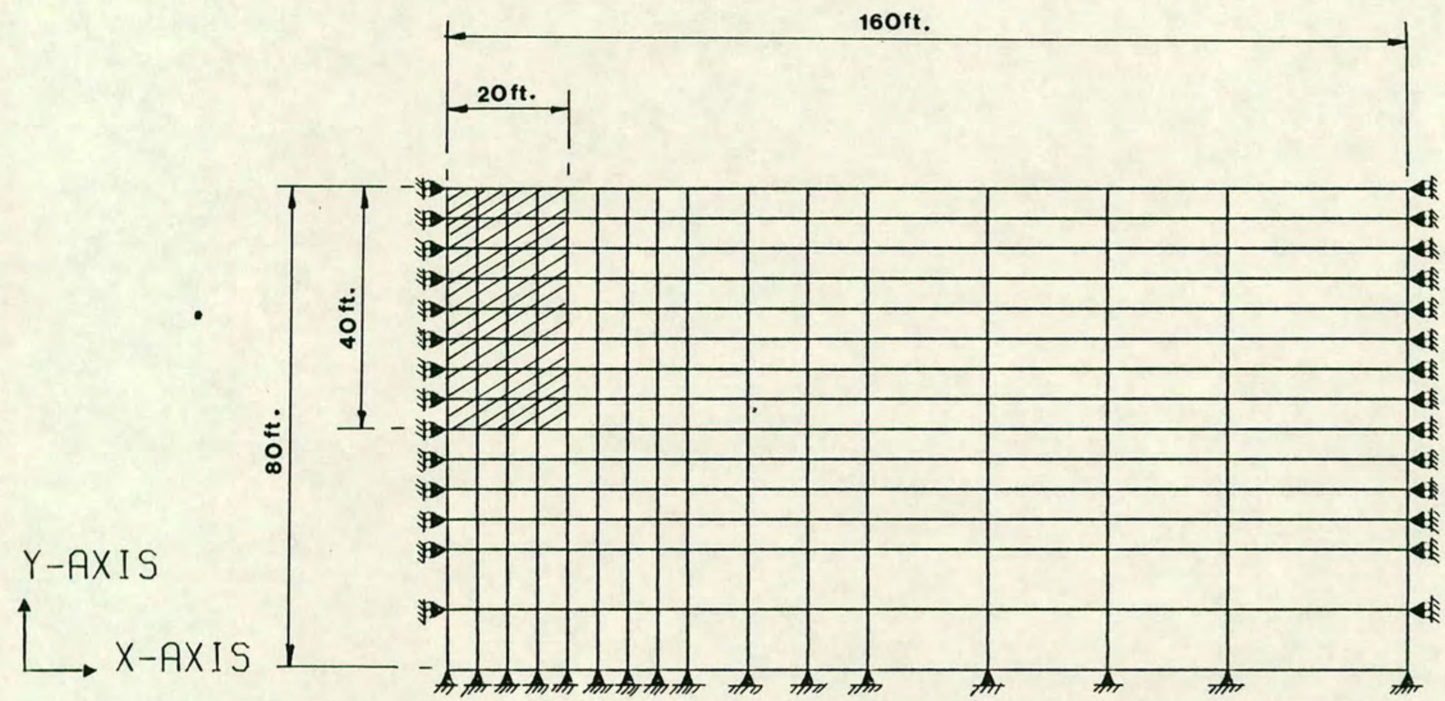


Figure 5.8

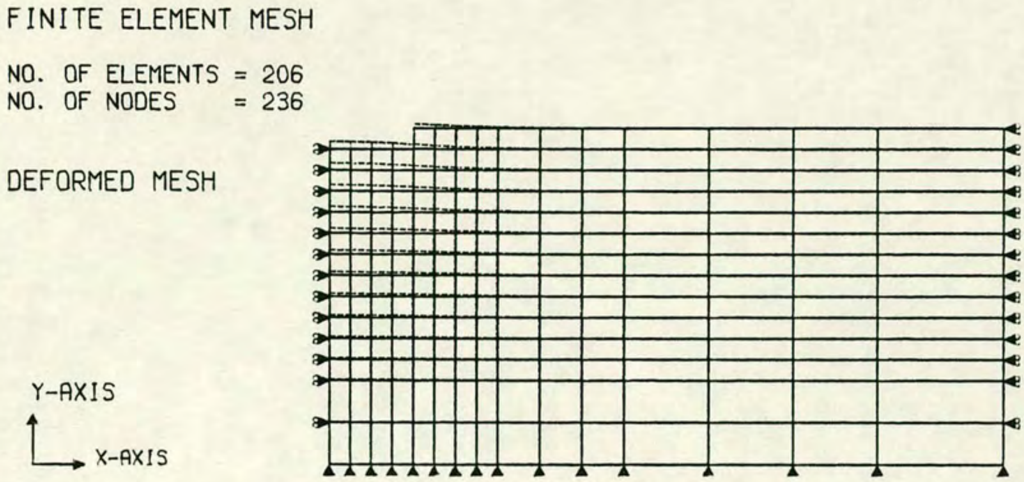


Figure 5.9(i)

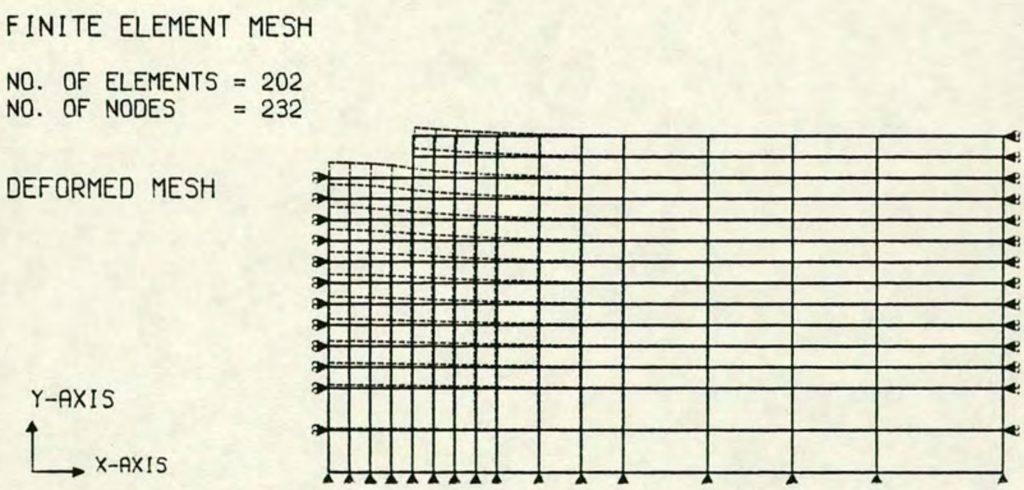


Figure 5.9(ii)

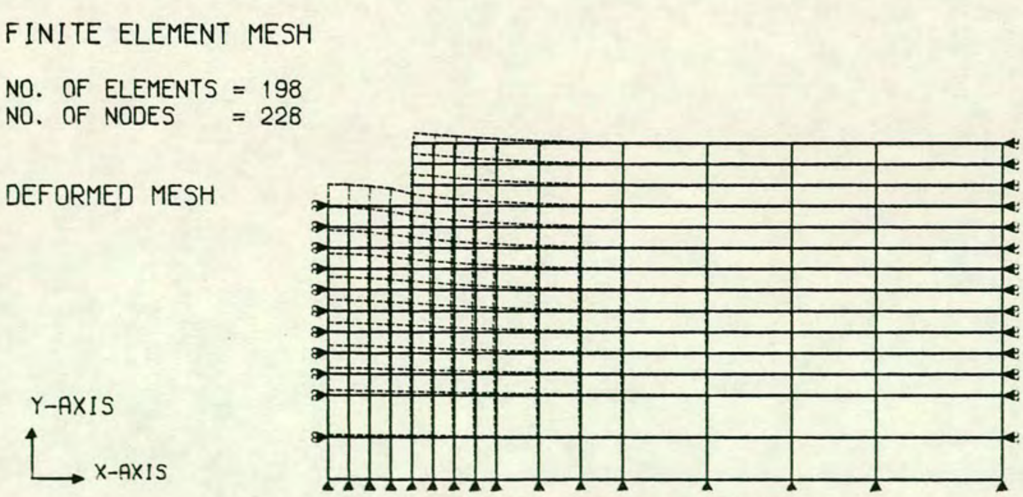


Figure 5.9(iii)

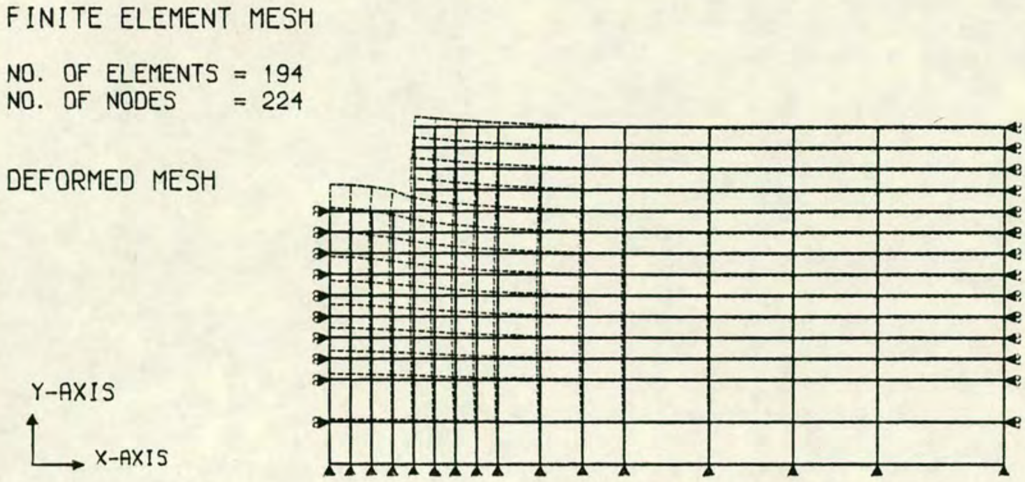


Figure 5.9(iv)

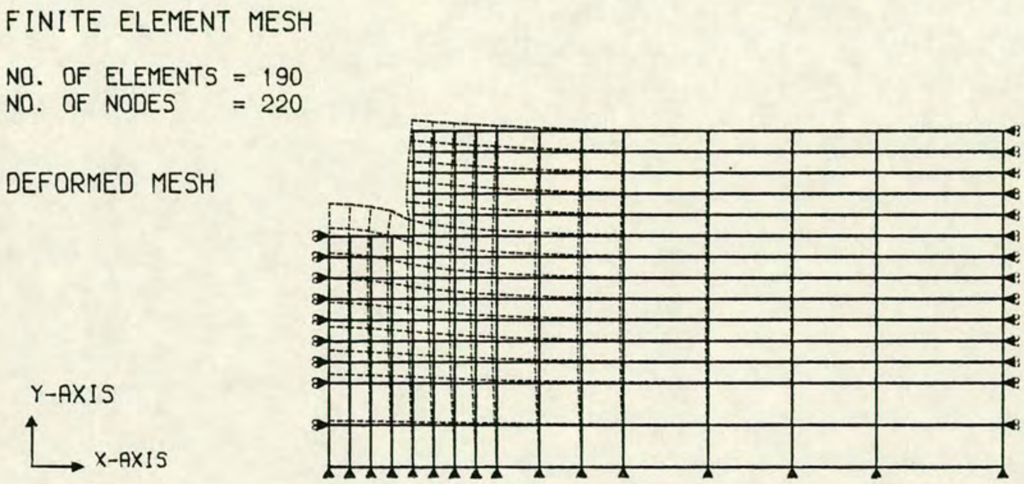


Figure 5.9(v)

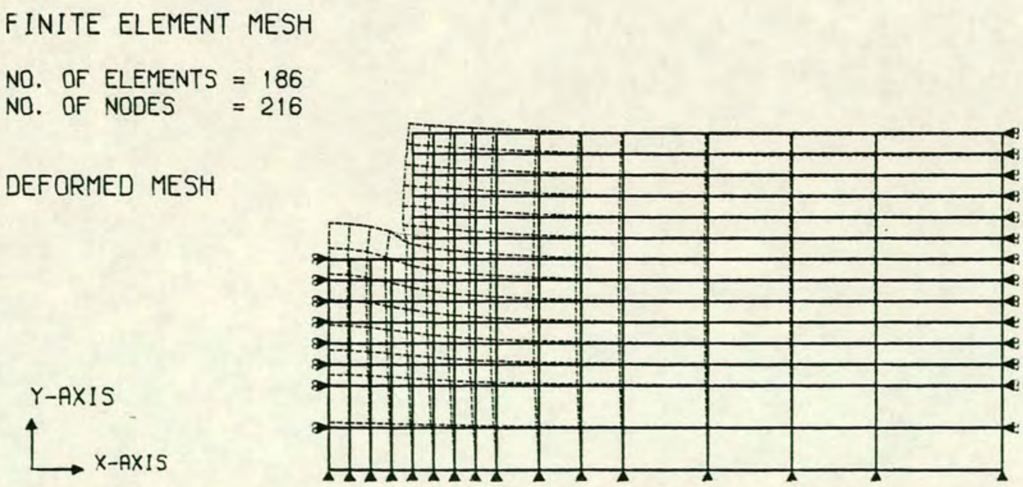


Figure 5.9 (vi)

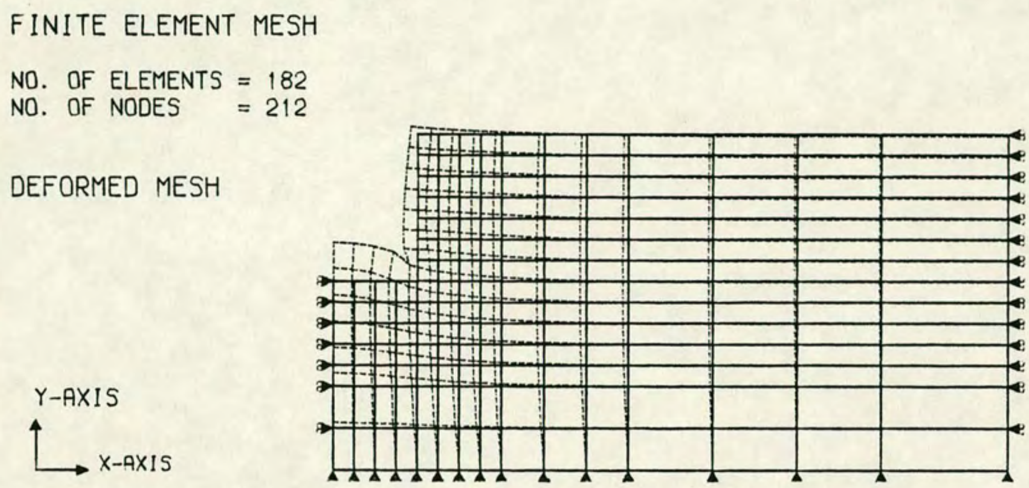


Figure 5.9 (vii)

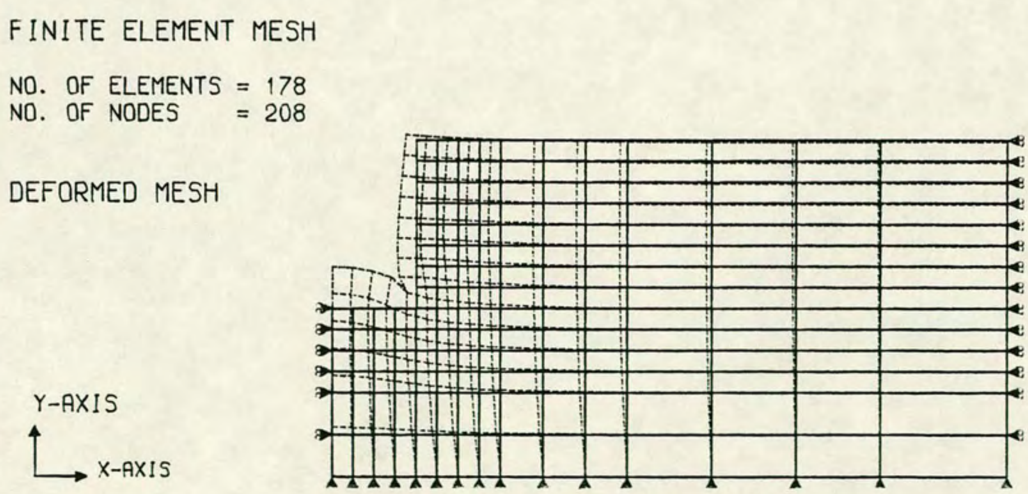


Figure 5.9 (viii)

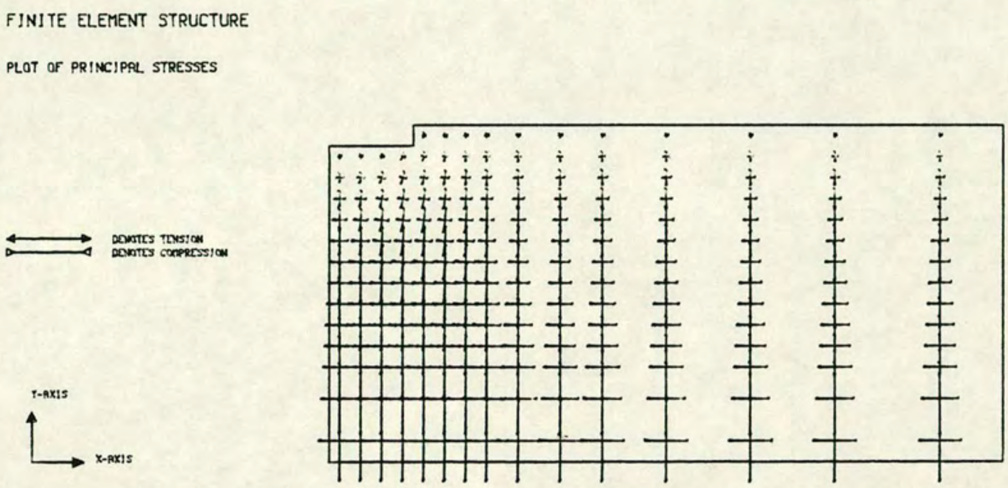


Figure 5.10(i)

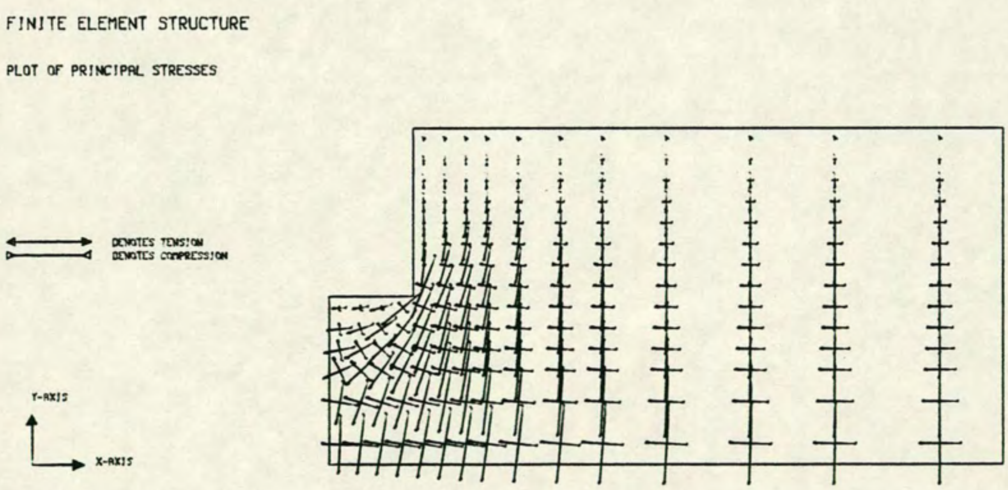


Figure 5.10(ii)

100,000 p.s.f., 100 p.c.f., 0.333 and 0.5 respectively.

The problem is modelled using the QUAD4 elements and the excavated area is shown hatched in figure 5.8. The excavation area is 40 ft. deep and 20 ft. wide. It is simulated by removal of a layer of elements of 5 ft. deep each at a time. This gives a total of eight stages of excavation. The initial analysis only involves the unit loads applied to the hatched elements of figure 5.8.

5.3.2.2 Analysis of each stage of excavation

At the first stage of the excavation, the elements to be removed are simulated by setting the elastic modulus to zero. The applied loads are calculated using equation (5.3). The displacements and stresses as a result of this applied loads on the original structure of figure 5.8 are obtained by superposition of the unit load analyses.

The theorems of structural variation are then used to obtain the incremental displacements and stresses. The total displacements and stresses are given by equation (5.4). The process is repeated for the next stage of excavation. It should be noted that the analysis is similar to that of the nonlinear solution techniques in Chapter 4.

The stresses and displacements are exact for each stage of the excavation using the theorems. Plots of deformed mesh superimposed on the original mesh are illustrated in figures 5.9(i) to (viii). The principal stresses at the first and last stage of the excavation is shown in figure 5.10. Figure 5.9 outlines the construction history of the excavation problem and figure 5.10 indicates the change in stresses from the first to the final stage of the excavation. The state of initial stress is greatly distorted in the vicinity of the excavation.

The CPU time for each reanalysis are tabulated in table 5.5.

	CPU time taken (seconds)	
Excavation stage (Figure)	Fresh analysis	Reanalysis by the theorems
0th. 5.8	--	23.91 (initial analysis)
1st. 5.9(i)	10.96+5.51=16.11	3.34
2nd. 5.9(ii)	10.96	3.46
3rd. 5.9(iii)	10.96	3.65
4th. 5.9(iv)	10.96	3.90
5th. 5.9(v)	10.96	4.22
6th. 5.9(vi)	10.96	4.62
7th. 5.9(vii)	10.96	5.11
8th. 5.9(viii)	10.96	5.70
Total CPU time Σ	92.83	57.91

Table 5.5 CPU time for reanalysis

For the analysis by the usual procedure the first stage takes 16.11 seconds which includes 5.15 seconds for reading data and details of the excavation sequence. Each subsequent analysis takes 10.96 seconds. Therefore the total CPU time taken is 5.15 + (8 X 10.96) = 92.83 seconds.

The time taken for the initial analysis of the original mesh shown in figure 5.8 is a large proportion of the total time. This analysis is for the unit loads at the nodes of each of the element to be removed in the excavation sequence. Nevertheless the total time taken using the theorems is less than the time for a series of fresh analyses. This is because the subsequent reanalysis using the theorems takes only a fraction of the time taken for a fresh analysis. Each reanalysis is referred to the original structural analysis. The theorems become more efficient as the number of excavation stages increases. The usual method of analysis[27] is to incorporate fictitious elements with small E values to simulate the excavation. This procedure may lead to numerical difficulties in the solution algorithm. No such difficulties were encountered using the theorems where the removal of the elements may be undertaken by setting $\alpha^e = -1$. The equilibrium equations using this value of α^e are well-conditioned and the results are exact.

5.3.3 Final comments

The theorems of structural variation should not be thought of as limited only to removal of elements as illustrated in the preceding examples. They may easily be used for variations in the thickness of the shear wall. Instead of using the elastic modulus in equation (3.1) the thickness, t may be substituted. Therefore the response of a shear wall with greater thickness at the base than at the upper storeys may be investigated. The effects of increasing the thickness near openings to reduce the local stresses may also be undertaken by the theorems. Any patterns of cut-outs may be considered using the theorems.

In the excavation problem, more complex construction stages may also be investigated. In addition the variations in material properties using equation (3.1) may be used to study the response of the excavation process. This is important because the material behaviour at the later stages of the excavation may be nonlinear. This may only be undertaken using the theorems when the new stiffness is a scalar multiple of the original stiffness.

5.4 Summary

The efficiency of the theorems of structural variation have been investigated using the formulations of Chapter 3. The technique of evaluating the condensed variation factors directly was shown to be the most efficient formulation. The theorems were also shown to be most efficient when applied to idealisations using higher order elements. The efficiency tests also indicate that the theorems will be particularly efficient for locally modified large structures having large semi-bandwidth. The solutions obtained were exact and no round-off errors or numerical difficulties were encountered.

The two examples clearly show that the theorems are an efficient and practical reanalysis technique for the analysis of a series of modified structures. The time taken for each reanalysis is only a fraction of the time for a fresh analysis. Hence it would a useful design tool to investigate many alternative designs by a trial and error approach based on experience and intuition. The theorems may also be incorporated as part of an automated design process using optimization techniques where many modified designs are required.

CHAPTER 6

EFFICIENCY OF THE THEOREMS OF GEOMETRIC VARIATION AND SOME POTENTIAL APPLICATIONS

6.1 Introduction

In this chapter the efficiency of the theorems of geometric variation for reanalysis is investigated. The tests were undertaken on some simple finite element structures. The efficiency was based on the comparison of the CPU times taken in using the theorems as a reanalysis technique with that for a complete or fresh analysis.

In addition two examples were illustrated as potential uses of the theorems of geometric variation. The first example is the reanalysis of a fillet tension bar in order to minimise the stress concentration factor. The second involved the analysis of an unconfined seepage flow problem.

6.2 Investigation of the efficiency

The formulation of the theorems of geometric variation given in matrix form in Chapter 3 were tested for efficiency. The elements considered in the reanalysis were the 3-node triangular and the 4-node isoparametric quadrilateral elements.

The first test involved the comparison of the efficiency between the CST and NST triangular elements using different formulations to form the matrix of compensation forces given by equation (3.52). The first type of formulation used only the NST elements to form the compensation forces from equation (3.32). In this formulation the edge extensions were evaluated from the nodal displacements due to the unit load analyses. The element edge forces before and after modification were obtained using the NST elements. The CST elements were used for the initial analysis. Using NST elements in this way is a direct analogy to the use of the theorems of geometric variation for truss structures. The second type of formulation involved in using the CST elements before modification and the NST elements after modification. Therefore in equation (3.32) the terms $f_{i,jh}\sin\theta_{ik}$ and $f_{i,jh}\cos\theta_{ik}$ were replaced by the nodal forces in the x and y directions. The third type of formulation used the CST elements before and after modification in where the compensation forces in equation (3.50) are given by the difference in the nodal forces in the x and y directions. For the QUAD4 elements the general formulation of equation (3.50) is used.

The previous tests described above are relative tests to enable a comparison of the efficiency of the various formulations to determine the matrix of compensation forces. The effect on the reanalysis efficiency of using different elements to idealise a problem will also be determined. A comparison with a complete or fresh analysis is required to determine the absolute efficiency of the theorems of geometric variation.

6.2.1 The structures chosen as benchmark tests

The test problem is a thin plate (in plane stress) with an elliptical hole. A quarter of the plate was analysed as shown in figure 6.1. The nodes along the circumference of the ellipse were moved such that the hole becomes a circular one. The plate was subjected to a tensile stress of 500 N/mm^2 in the x-direction. Its elastic modulus was 2 X 10^6 N/mm^2, the Poisson's ratio 0.25 and the plate was of unit thickness. The dimensions of the problem are as shown in figure 6.1. The elliptical hole was modified to a circular one of radius 17.78 mm.

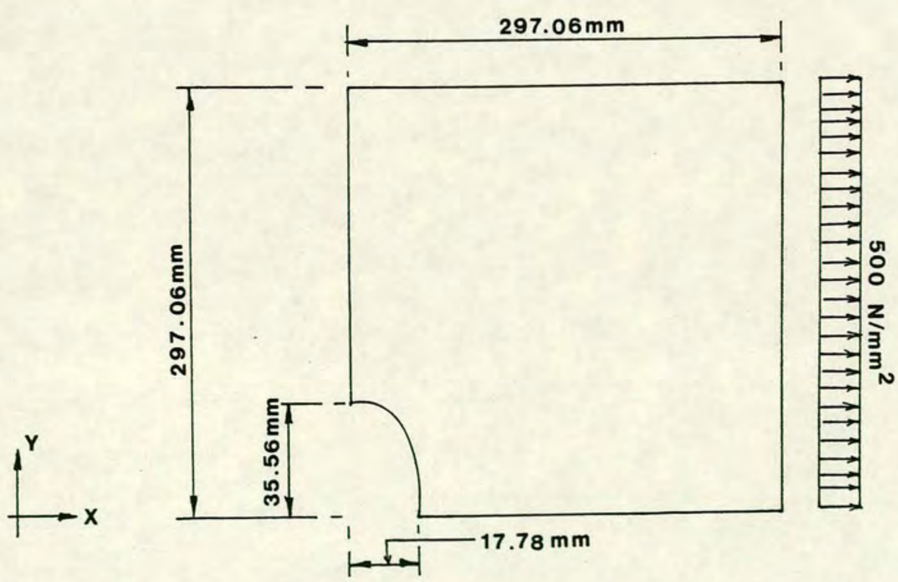


Figure 6.1 Thin plate in plane stress

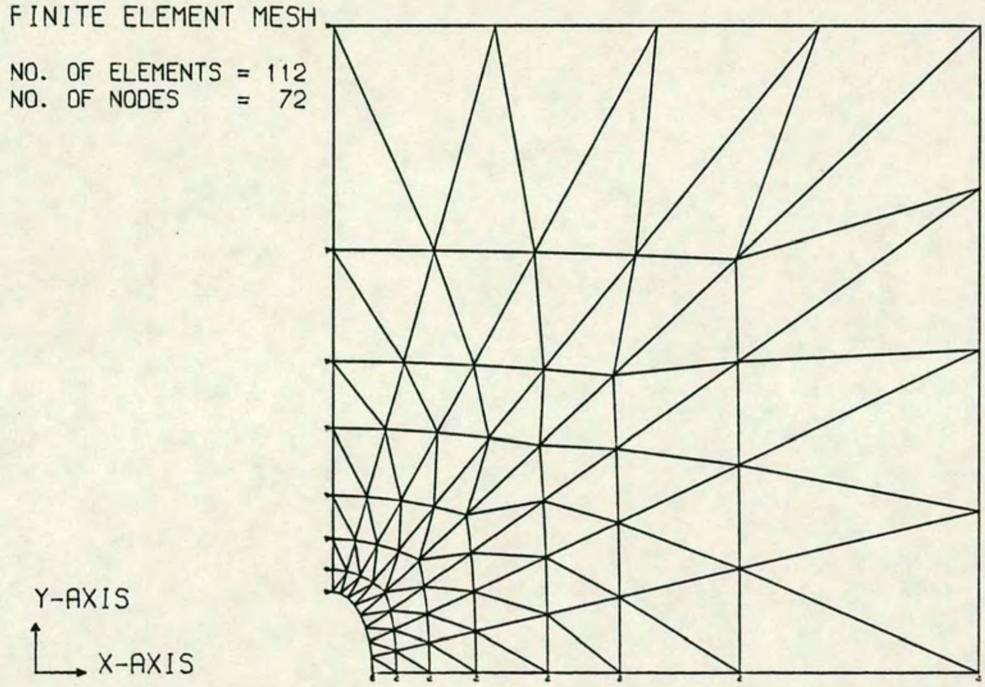


Figure 6.2(i)

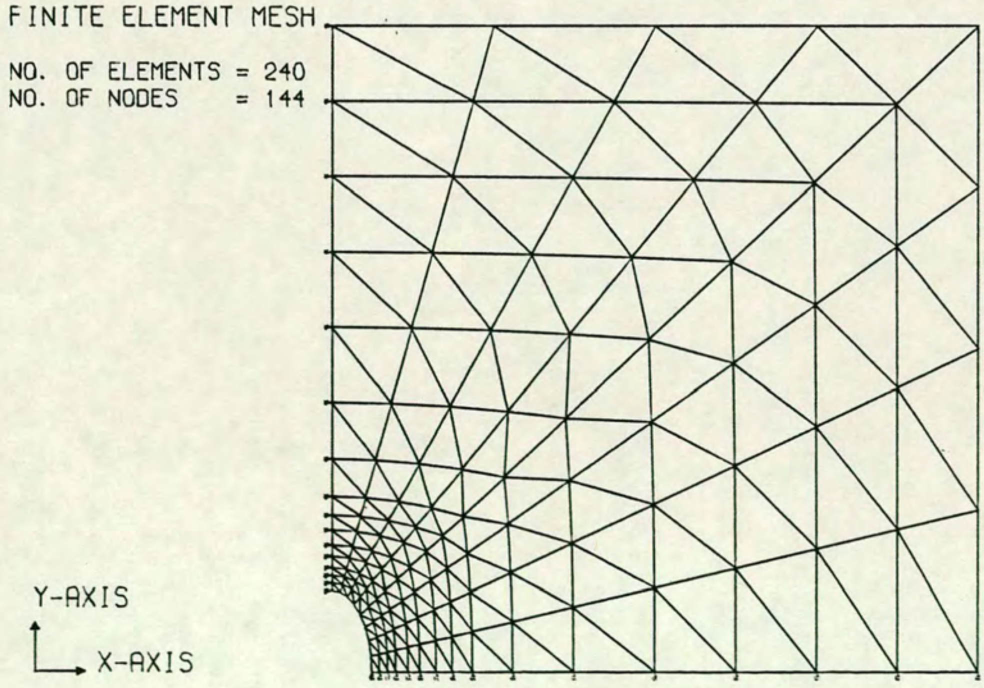


Figure 6.2(ii)

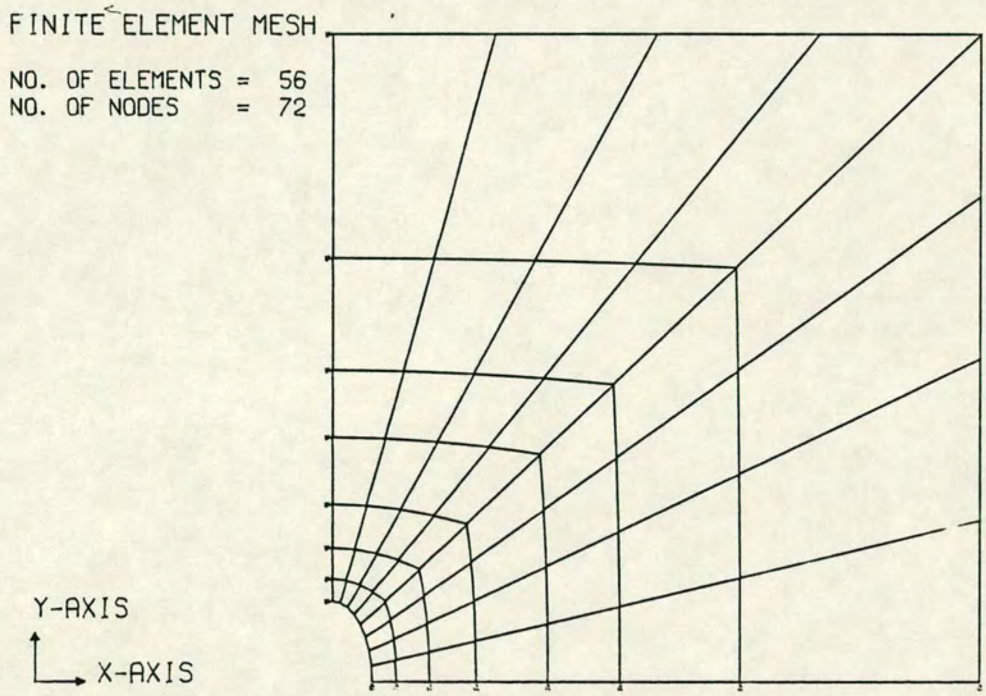


Figure 6.3(i)

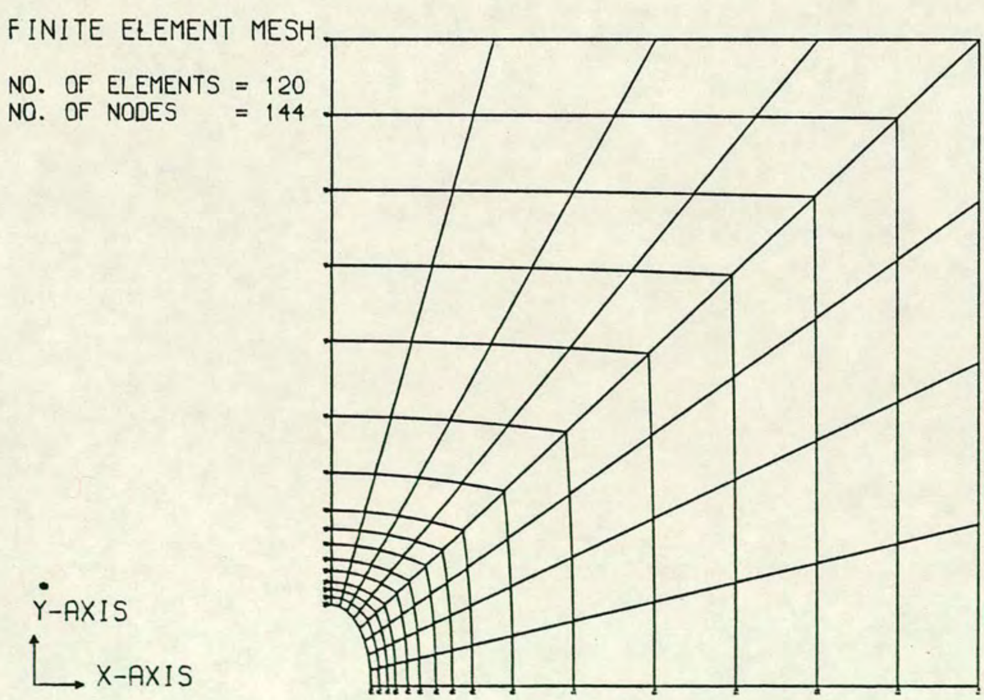


Figure 6.3(ii)

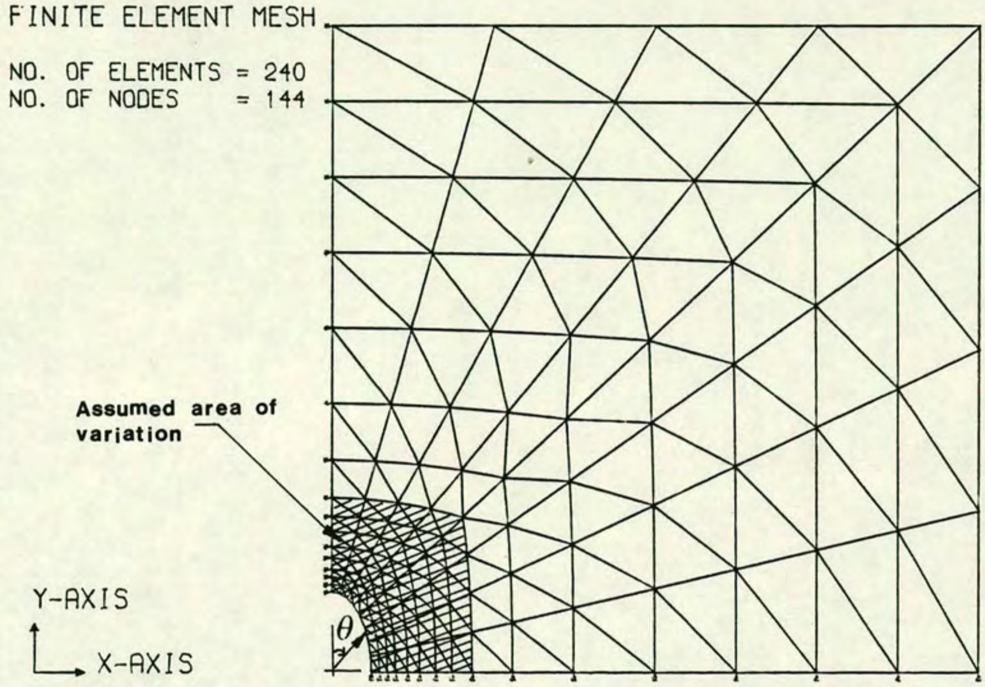


Figure 6.4(i) Original mesh

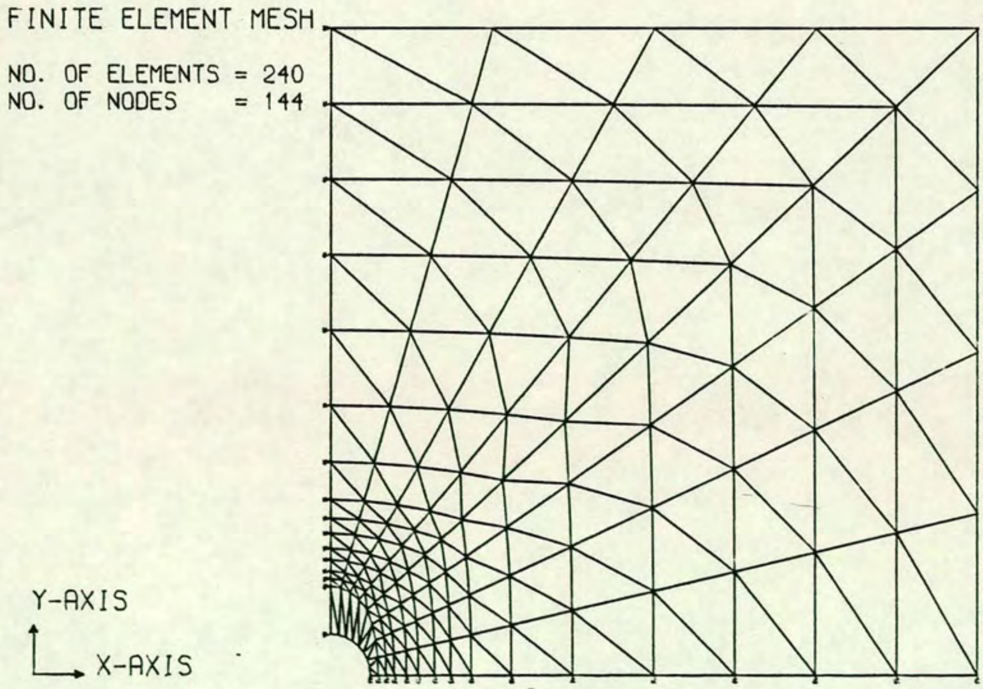


Figure 6.4(ii) 1st. layer

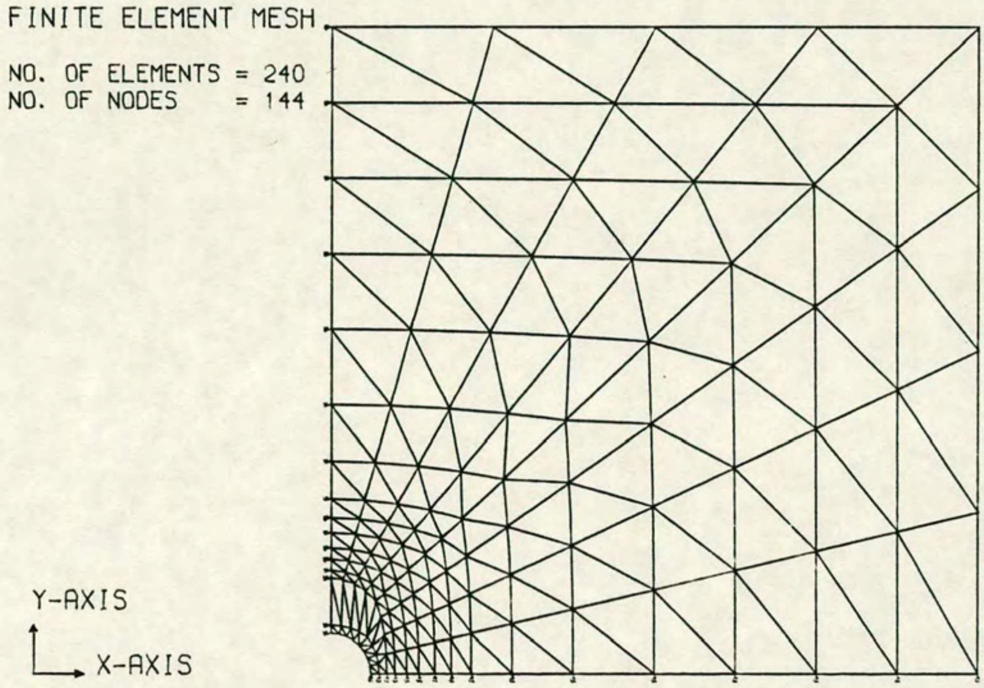


Figure 6.4(iii) 2nd. layer

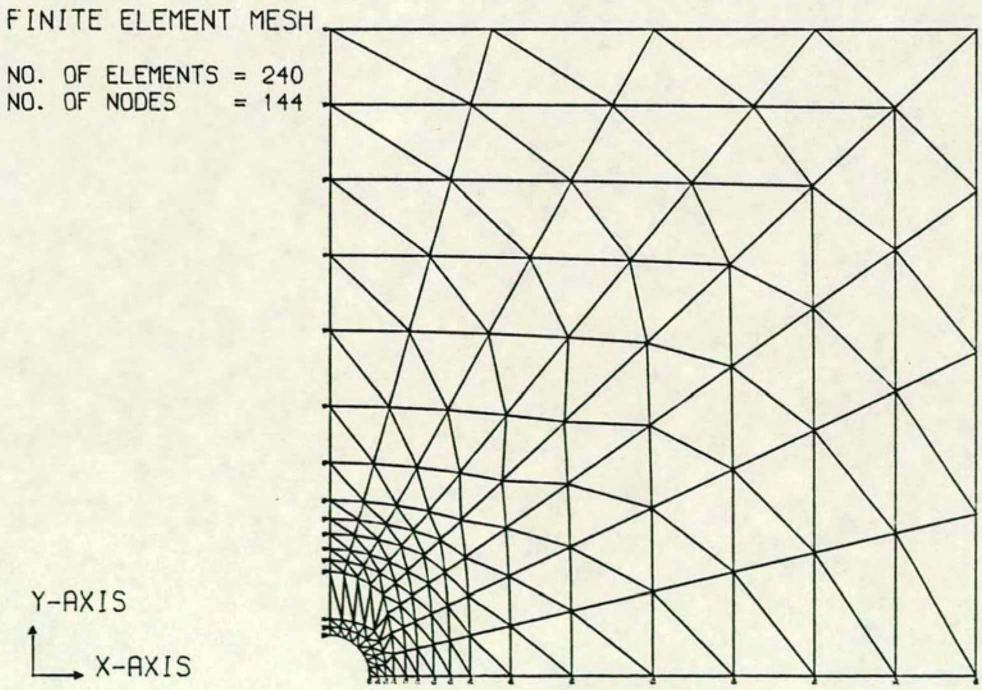


Figure 6.4(iv) 3rd. layer

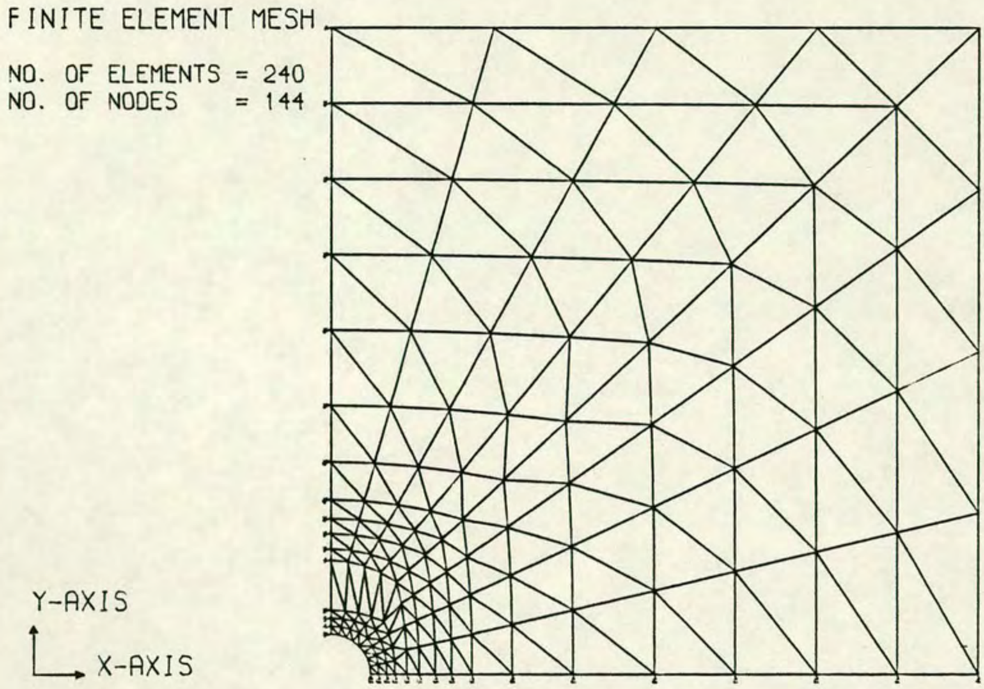


Figure 6.4(v) 4th. layer

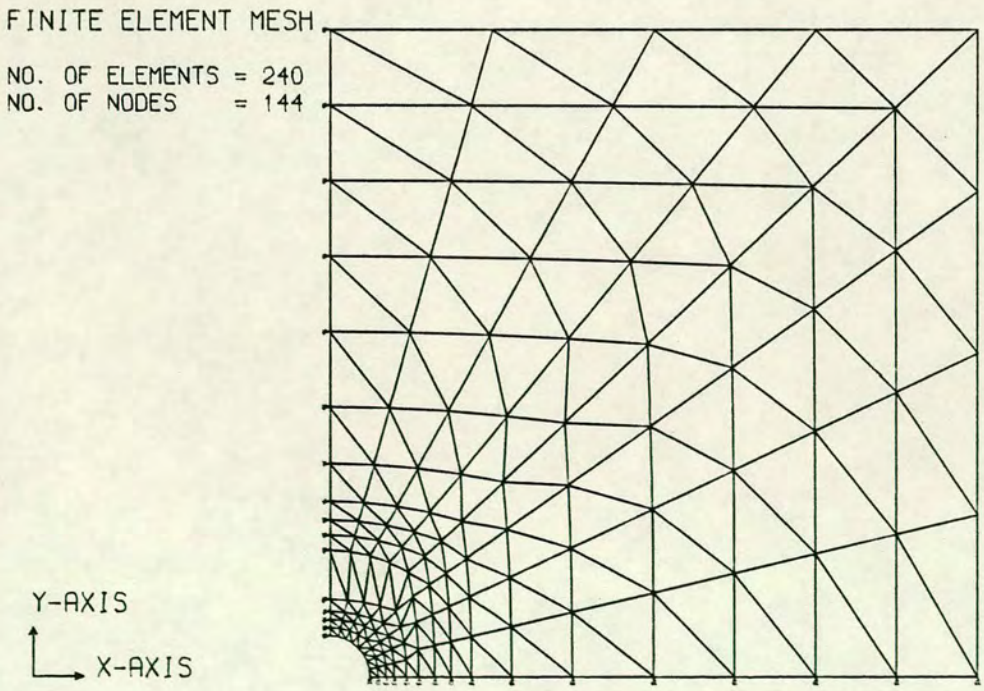


Figure 6.4(vi) 5th. layer

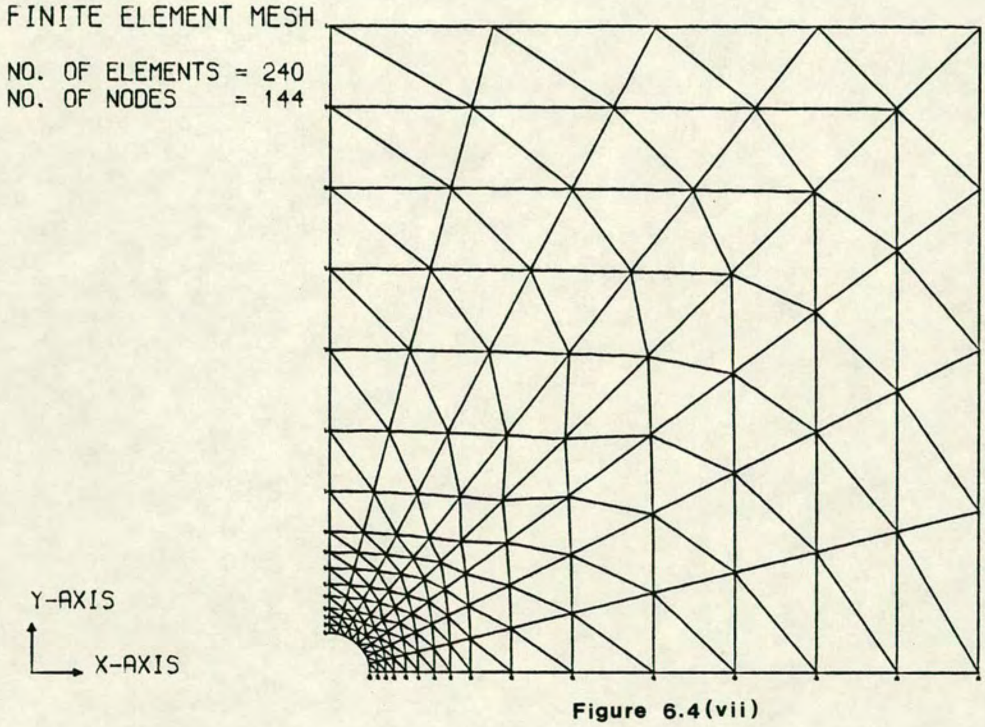


Figure 6.4(vii)

For the triangular elements the thin plate was divided into two meshes; a coarse and a fine mesh as shown in figures 6.2(i) and (ii) respectively. Similarly for the QUAD4 elements shown in figures 6.3(i) and (ii) where two triangular elements were taken as equivalent to one QUAD4 element. The type of elements used in the reanalysis, number of elements, nodes and semi-bandwidth for each structure are tabulated in table 6.1.

Structure (Figure)	Type of element used	No. of elements	No. of nodes	Semi-bandwidth
6.2(i)	CST and NST	112	72	20
6.2(ii)	CST and NST	240	144	36
6.3(i)	QUAD4	56	72	20
6.3(ii)	QUAD4	120	144	36

Table 6.1 Structures tested

The nodes along the boundary of the elliptical hole were varied one at a time, to make the circular hole shown in figure 6.4(ii). These nodes were the first layer of nodes varied to change the boundary of the problem. Obviously the distorted mesh near a region of high stress gradients should be avoided. Therefore the nodes along the second layer were varied one at a time until the structure in figure 6.4(iii) was obtained. This process was continued for the third layer in figure 6.4(iv) and so on. The CPU time taken for the number of nodes varied was measured.

6.2.2 The CPU time taken for the number of nodes varied

The graphs of CPU times against the number of nodes varied are plotted in figures 6.5(i) and (ii) for the coarse and fine meshes respectively. The type of formulations used for the compensation forces for each mesh are tabulated in table 6.2.

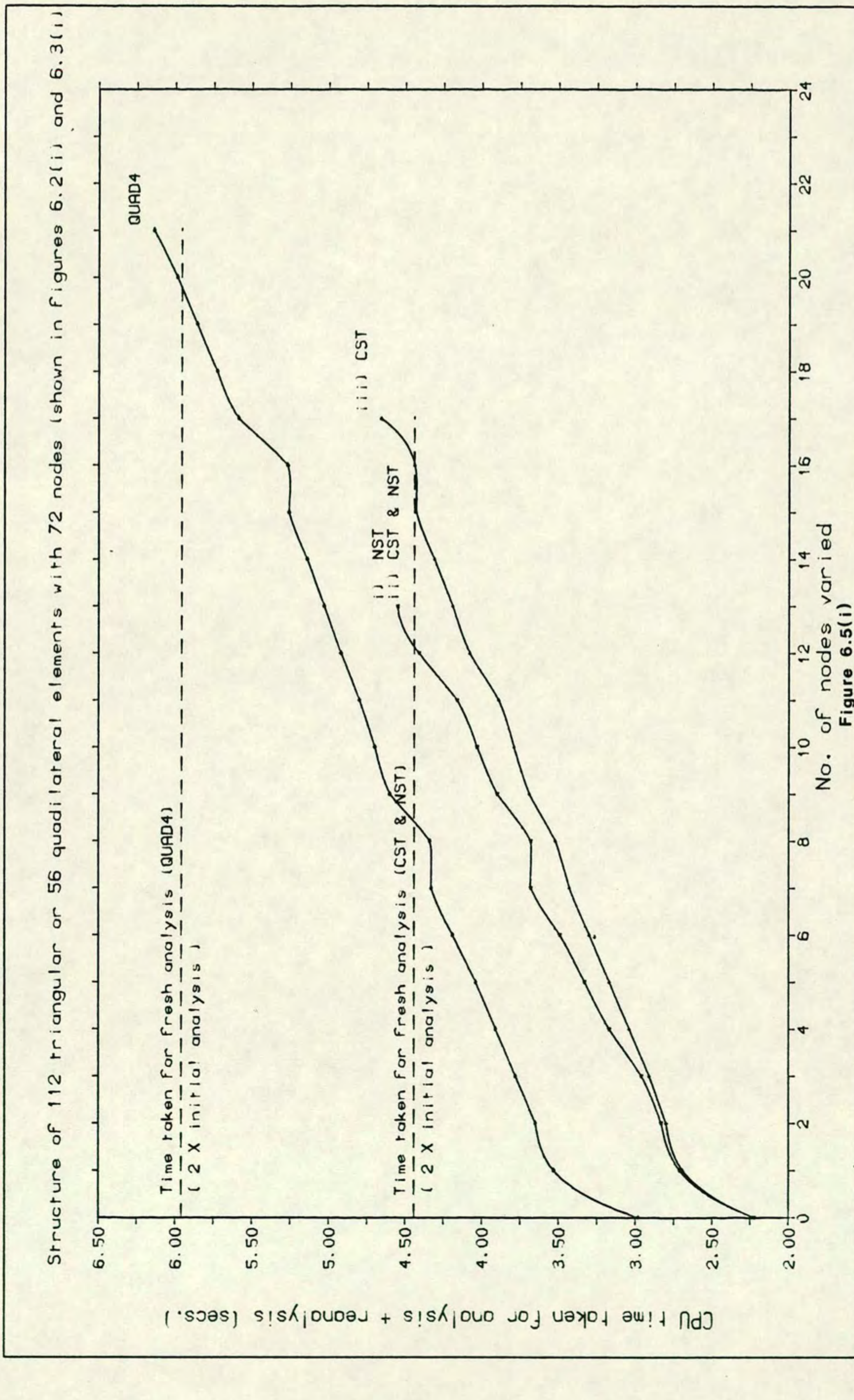

Structure of 112 triangular or 56 quadilateral elements with 72 nodes (shown in Figures 6.2(i) and 6.3(i)
CPU time taken for analysis + reanalysis (secs.)
No. of nodes varied
Time taken for fresh analysis (QUAD4)
(2 X initial analysis)
Time taken for fresh analysis (CST & NST)
(2 X initial analysis)
QUAD4
i) NST
ii) CST & NST
iii) CST
6.50
6.00
5.50
5.00
4.50
4.00
3.50
3.00
2.50
2.00
0
2
4
6
8
10
12
14
16
18
20
22
24

Figure 6.5(i)

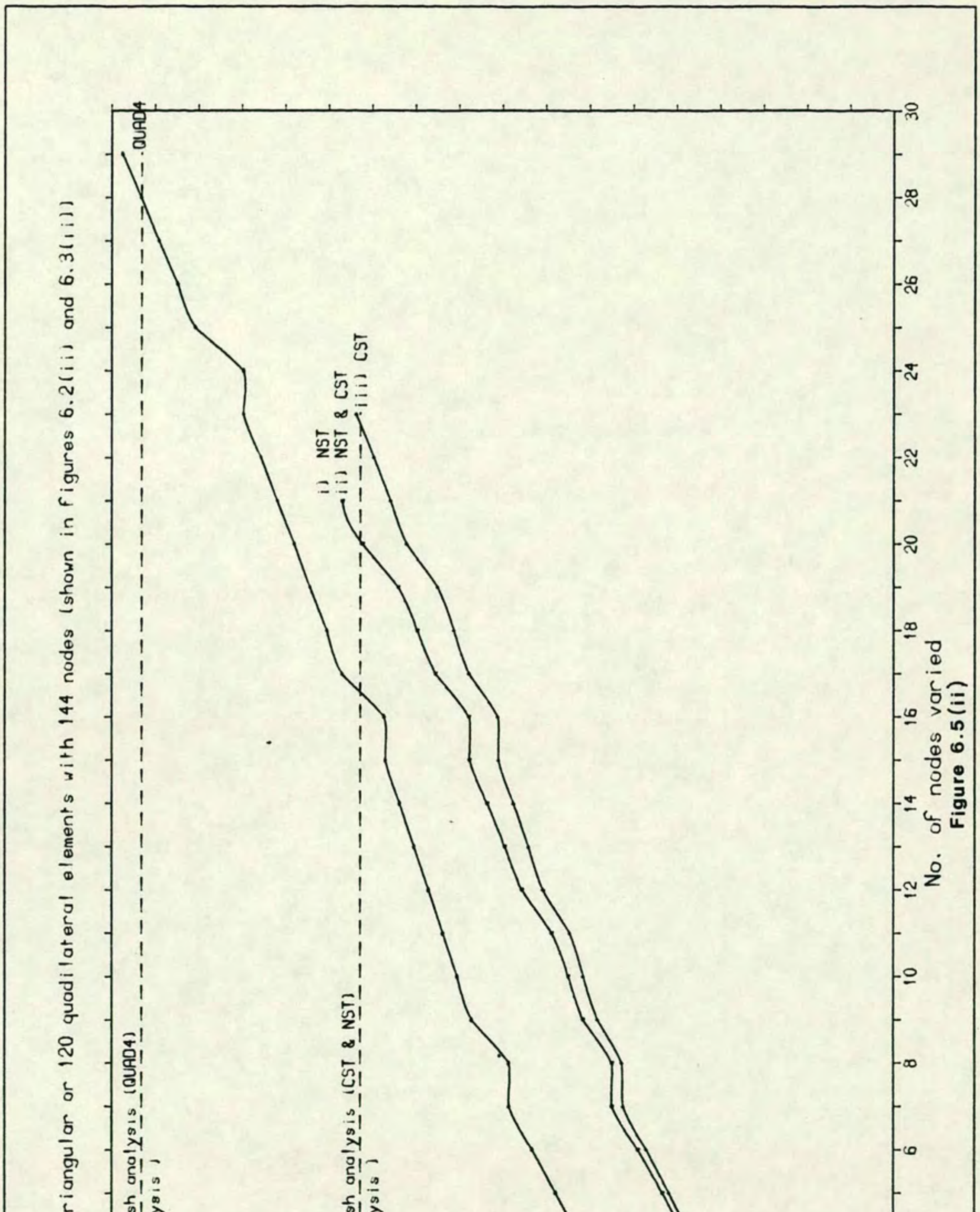

Structure of 240 triangular or 120 quadrilateral elements with 144 nodes (shown in figures 6.2(ii) and 6.3(iii))
Time taken for fresh analysis (QUAD4)
(2 X initial analysis)
QUAD4
Time taken for fresh analysis (CST & NST)
(2 X initial analysis)
i) NST
ii) NST & CST
iii) CST
CPU time taken for analysis + reanalysis (secs.)
19.50
18.17
16.83
15.50
14.17
12.83
11.50
10.17
8.83
7.50
0
2
4
6
8
10
12
14
16
18
20
22
24
26
28
30
No. of nodes varied

Figure 6.5 (ii)

Structure (Figure)	Type of element used	Formulation used for compensation forces
6.2(i),(ii)	(i) NST	Equation (3.32)
6.2(i),(ii)	(ii) CST & NST	Equation (3.32)†
6.2(i),(ii)	(iii) CST	Equation (3.50)
6.3(i),(ii)	QUAD4	Equation (3.50)

Table 6.2 Formulations used in the reanalysis

† Note that in equation (3.32) the terms $f_{i,jh}\sin\theta_{ik}$ and $f_{i,jh}\cos\theta_{ik}$ are replaced by the nodal forces in the x and y directions.

The graphs may be used for comparison of the relative efficiency of the type of formulations and elements used. For the absolute efficiency a comparison is made with the CPU time taken for a fresh or a complete analysis. It is twice the time taken for a single analysis. This is shown by the horizontal 'broken' lines in the graphs. The time taken for a single analysis is when the number of nodes varied is zero.

6.2.3 Maximum percentage of nodes varied efficiently

This provides a measure of efficiency between the triangular and quadrilateral elements. It also indicates how many nodes may be varied as the structure increases in size (in this case a finer mesh). The maximum percentage of nodes varied efficiently is define as:

$$\text{Max. \% of nodes varied efficiently} = \frac{\text{Max. no. of nodes varied efficiently}}{\text{Total no. of nodes}} \times 100\,\% \qquad (6.1)$$

Table 6.3 represents the maximum percentage of nodes varied efficiently for each type of formulation and structure used.

	Structural idealisation (Max. % of nodes varied efficiently)	
Type of elements and formulation	Figure 6.2(i) (72 nodes)	Figure 6.2(ii) (144 nodes)
(i) NST Equation (3.32)	16.67%	13.19%
(ii) CST & NST Equation (3.32)†	16.67%	13.19%
(iii) CST Equation (3.50)	22.22%	15.23%
	Figure 6.3(i) (72 nodes)	Figure 6.3(ii) (144 nodes)
QUAD4 Equation (3.50)	26.34%	18.75%

Table 6.3 Maximum percentage of nodes varied efficiently

6.2.4 Discussion of results

The efficiency of evaluating the compensation forces using NST elements by equation (3.32) or (3.32)† are the same as indicated in the graphs of figure 6.5 and table 6.3. The formulation using equation (3.50) for the CST elements is the most efficient. This is because when using the NST elements, the edge forces must be resolved to obtain the nodal forces. The calling routines for the sine and cosine functions in a computer program is computationally inefficient. The QUAD4 elements is the most efficient of all the formulations since the number of nodes that may be varied efficiently is the greatest. This is because for a varied node the number of affected elements is generally reduced by half. Hence the modified element stiffnesses are formed only for one element in comparison to two CST elements. In addition, the number of compensation forces calculated is generally less when using an equivalent QUAD4 element idealisation.

In table 6.3, the maximum percentage of nodes varied efficiently drops as the number of elements in the finite element idealisation becomes larger. Therefore the use of higher order elements would be advantageous since the number of elements required to model a given problem is generally less than using simpler elements. This suggests that the theorems are only efficient for locally modified structures.

It is quite clear from the matrix formulations of Chapter 3 that the main CPU time taken for the reanalysis is for:

1. The reduction of the overall stiffness matrix and the multiple loading cases of equation (3.47). As each node is varied the number of unit loads at the 'a' affected nodes increases.

2. The solution for the scale factors using equation (3.52) takes a large proportion of the total CPU time. The matrix of compensation forces is full and unsymmetric and therefore this is the most inefficient part of the reanalysis technique. As in Chapter 5 partial pivoting of the matrix is not required since the solutions obtained do not suffer from any round-off errors and numerical difficulties were not encountered.

3. The recalculation of the modified element stiffness matrices using equation (3.49c). As the number of varied nodes increases the number of affected elements also increases. The modified stiffness is not a scalar multiple of the original stiffness as required in the theorems of structural variation.

The size of the semi-bandwidth of the structure does not affect the reanalysis. Therefore the technique will be particularly effective for large locally modified structures with a large semi-bandwidth.

6.3 Investigation of the accuracy and comparison of the CPU time taken for each layer varied

The accuracy of the idealisations generated by varying the positions of the layers may be monitored by a comparison of the displacements and stress concentration factors at the circumferential nodes of the hole. The affected areas were assumed to be the elements shown hatched in figure 6.4(i). For each layer of nodes that was varied, the CPU time taken was measured. The initial analysis only involves unit load analyses of the assumed affected region and the nodes where there are applied loads. Subsequent reanalysis uses the theorems of geometric variation to find the response as each layer is varied. This test is only carried out for the fine mesh of triangular elements of figure 6.2(ii). In addition, the results of the meshes that were generated was compared with that of a mesh with an improvement in the grading of the elements, shown in figure 6.4(vii).

6.3.1 Graph of displacement

The displacements in the x-direction at the circumferential nodes of the hole are plotted against the angle, θ. The angle θ is defined in a clockwise direction as indicated in figure 6.4(i).

The first five layers that were varied are plotted in figure 6.6(i). As the number of layers increases, the values begin to converge. For the first layer, the results are inaccurate because of the severe distortion of the element idealisation as illustrated in figure 6.4(ii). Reasonable accuracy was achieved for the fifth layer, in comparison to the results using the mesh (with an improvement of the grading of the elements) of figure 6.4(vii). The layers that were being varied are as shown in figures 6.4(ii) to (vi).

6.3.2 Graph of stress concentration factor

This provides another measure of accuracy. The stress concentration factor[42] was defined as:

$$K_t = \sigma_p/\sigma_s \qquad (6.2)$$

where:

K_t = stress concentration factor;

σ_p = maximum principal stress; and

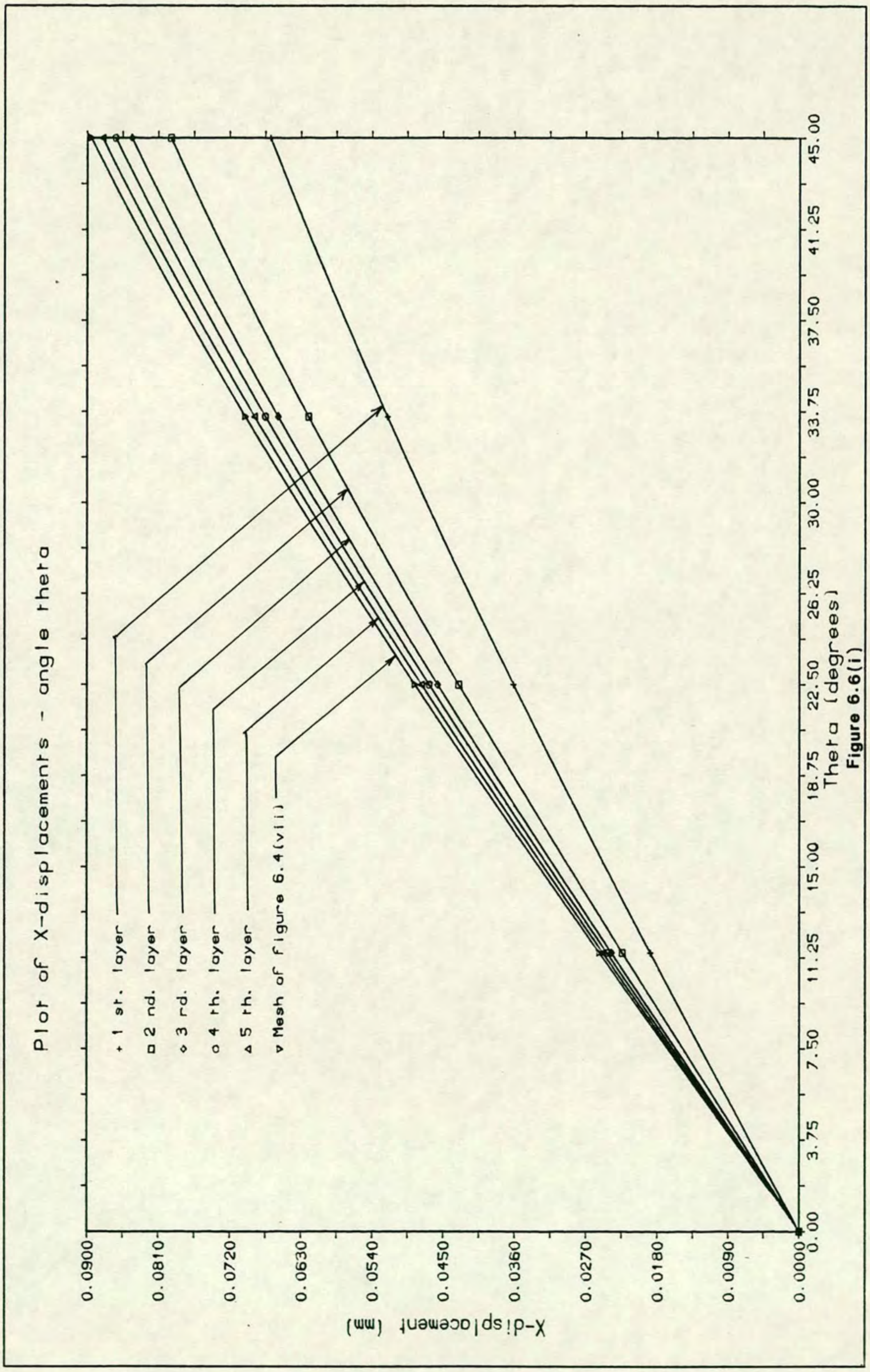
σ_s = some standard stress.

The stresses at the circumferential nodes were simply found by averaging the stresses of the elements connected to that node. Then the principal stresses may be evaluated. The standard stress is the applied stress of 500 N/mm^2.

The graph for the five layers that were varied are plotted in figure 6.6(ii). It is again clear that for the first layer the results are unreliable because the elements are severely distorted. As the number of layers increases, the mesh begins to give a better representation of the stress concentrations. This may be compared with the results using the mesh of figure 6.4(vii). However the results of the stress concentrations for the fifth layer is not as good as for the displacements. The value for the photoelastic analysis of Peterson[114] at $\theta=0^{\circ}$ is $K_t=2.80$. This is near to the value obtained for the fifth layer of $K_t=2.84$ at $\theta=0^{\circ}$.

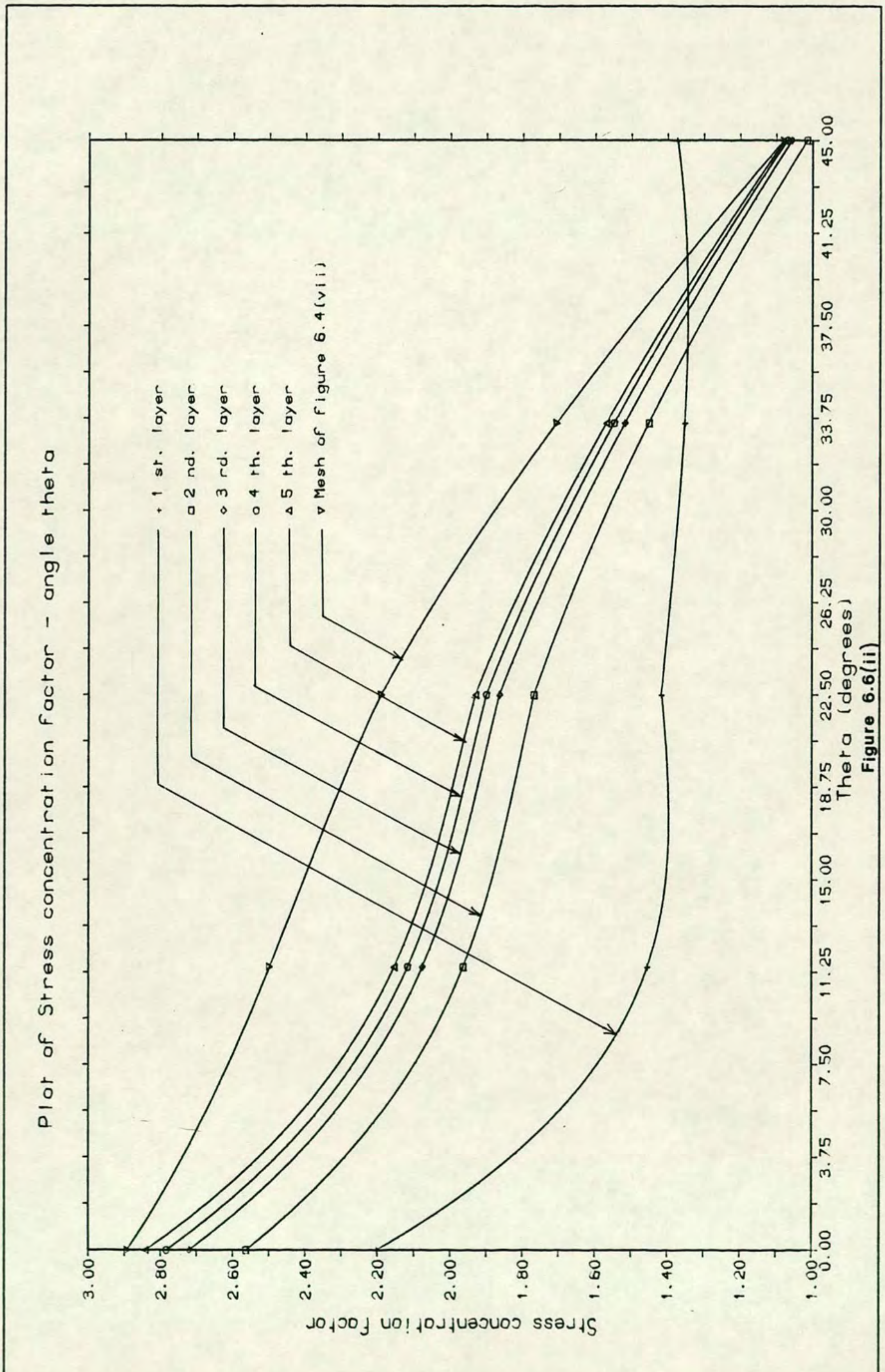
6.3.3 The CPU time taken for the number of layers varied

The comparison of the CPU times taken are tabulated in table 6.4.



Plot of X-displacements - angle theta
+ 1 st. layer
□ 2 nd. layer
◇ 3 rd. layer
○ 4 th. layer
△ 5 th. layer
▽ Mesh of figure 6.4(vii)
X-displacement (mm)
0.0000
0.0090
0.0180
0.0270
0.0360
0.0450
0.0540
0.0630
0.0720
0.0810
0.0900
0.00
3.75
7.50
11.25
15.00
18.75
22.50
26.25
30.00
33.75
37.50
41.25
45.00
Theta (degrees)

Figure 6.6(i)



Plot of Stress concentration Factor - angle theta
+ 1 st. layer
□ 2 nd. layer
◇ 3 rd. layer
○ 4 th. layer
△ 5 th. layer
▽ Mesh of Figure 6.4(vii)
Stress concentration Factor
Theta (degrees)
1.00
1.20
1.40
1.60
1.80
2.00
2.20
2.40
2.60
2.80
3.00
0.00
3.75
7.50
11.25
15.00
18.75
22.50
26.25
30.00
33.75
37.50
41.25
45.00

Figure 6.6(ii)

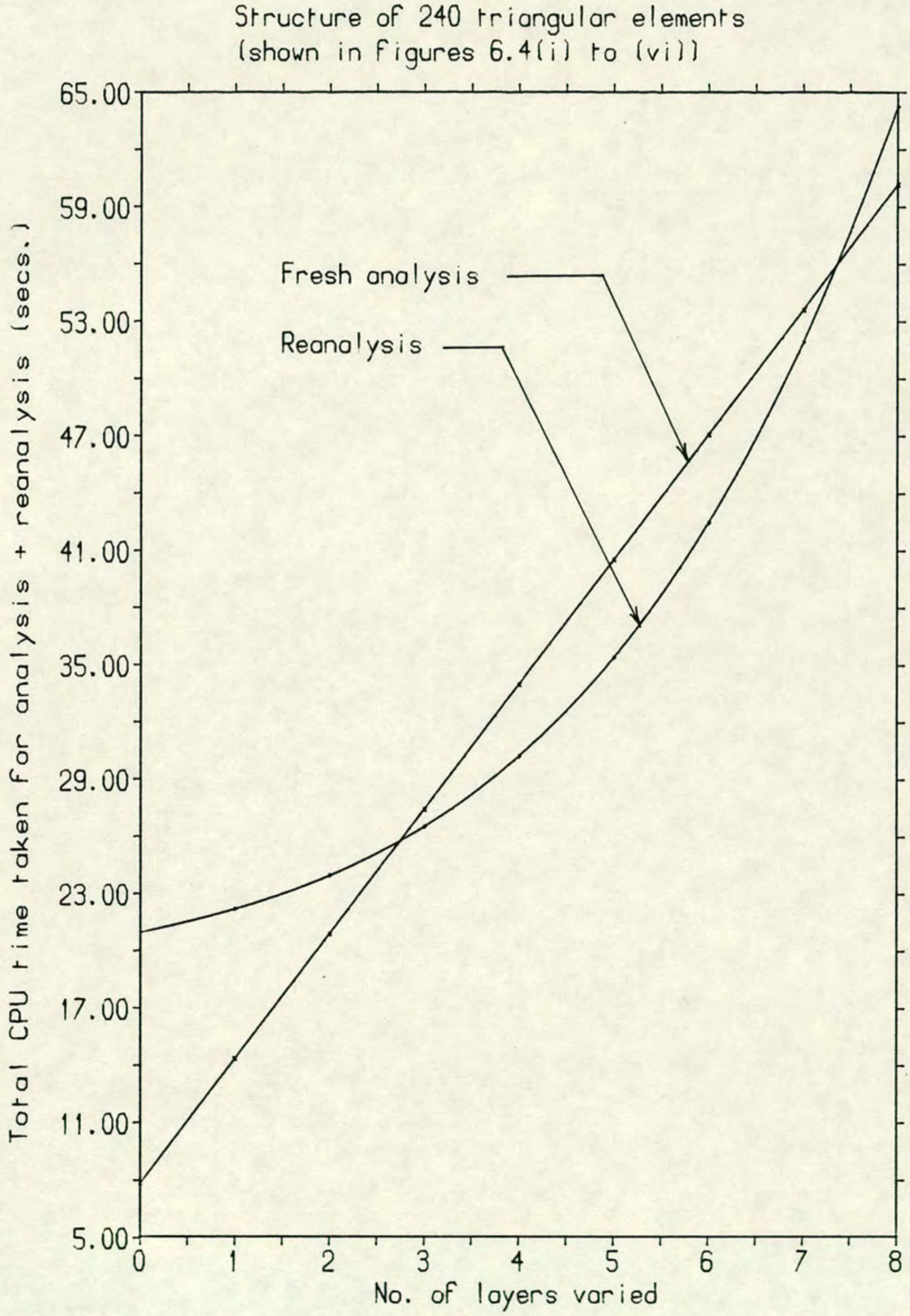


Figure 6.6(iii)

	CPU time taken (seconds)	
Structure (Figure)	Fresh analysis	Reanalysis by the theorems
0th. layer 6.4(i)	7.83	21.00 (Initial analysis)
1st. layer 6.4(ii)	6.54	1.23
2nd. layer 6.4(iii)	6.54	1.76
3rd. layer 6.4(iv)	6.54	2.57
4th. layer 6.4(v)	6.54	3.68
5th. layer 6.4(vi)	6.54	5.17
Total CPU time Σ	40.53	35.41
6th. layer	6.54	7.07
7th. layer	6.54	9.44
8th. layer	6.54	12.32

Table 6.4 CPU time for reanalysis

In the table the CPU time of 7.83 seconds includes the reading in of data concerning the mesh changes. The analyses of the modified structure is accomplished by reforming the structure stiffness matrix using the stored element stiffness matrices together with the recalculated matrices for the modified elements. This is a series of fresh analyses as described in Chapter 5.

For the reanalysis by the theorems of geometric variation, the initial analysis involves the unit loads at the assumed affected region of figure 6.4(i) (a total of 162 unit load analyses) in addition to the unit loads where there are applied loadings (10 unit load analyses). The subsequent reanalyses use the theorems to find the response of the modified structure. Each reanalysis is referred to the original structural analysis.

As shown in the table the reanalysis by the theorems of geometric variation is very efficient. It is inefficient for less than three modifications because the initial analysis of figure 6.4(i) requires a large proportion of the total CPU time. This is because unit loads must be applied at the (a+u) nodes of the original structure. However the subsequent reanalysis is only a fraction of the initial or the fresh analysis. Gradually it becomes more efficient as the number of layer increases.

The time for the reanalysis of the sixth layer by the theorems is 7.07 seconds. This time is greater than for a fresh analysis. Hence the efficiency at this stage begins to drop. This is because the reduction of equation (3.52) for the scale factors begins to take a large proportion of the total CPU time. This matrix as mentioned earlier is full and unsymmetric and therefore requires considerable computational effort to reduce when the number of varied nodes are large. This may be clearly observed in the graph of figure 6.6(iii). The graph is a plot of the total CPU time against the number of layers that were varied. For a few varied layers (less than three) the theorems are inefficient. The reanalysis is also inefficient when the number of varied layers is greater than seven. Obviously for an efficient reanalysis, a balance must be made in deciding the optimum number of reanalyses required. This in turn will be dependent on the type of problem under consideration as well as on the experience of the analyst. In general the theorems of geometric variation are only efficient for large structures requiring a small number of nodes to be varied.

6.4 Some applications

The potential uses of the theorems of geometric variation were investigated by considering two examples. The first one involved in finding the optimized shape of a fillet bar in tension such that the stress concentration was reduced. Optimization techniques were not used to find the optimized shape; a trial and error procedure was used instead. The aim here was to demonstrate the effectiveness and efficiency of the theorems. The second example involved the analysis of unconfined seepage flow where the location of the free surface is not known initially. Although this is not a structural problem, this particular example will show that the theorems have wider implications.

6.4.1 Shape optimization

The shape of a bar in tension where transition of one size to another is a common engineering problem. The change in transition results in regions of higher stresses than those normally applied. This example is a fillet tension bar analysed by Francavilla et.al.[42]. Photoelastic analysis of this problem is also provided by Peterson[114]. The analysis in reference [42] involves in finding the optimized shape so that the stress concentration around a local boundary was minimised. However in this case, the shape of the variable boundary will not be found by an optimizing technique as outlined by Francavilla et.al.[42]. A trial and error approach as discussed later was used, to illustrate the efficiency of the theorems as a reanalysis technique. The trial and error approach is usually used by engineers where optimization techniques are an unfamiliar tool.

6.4.1.1 Initial analysis

The fillet tension bar is in plane stress as shown in figure 6.7 for half the bar.

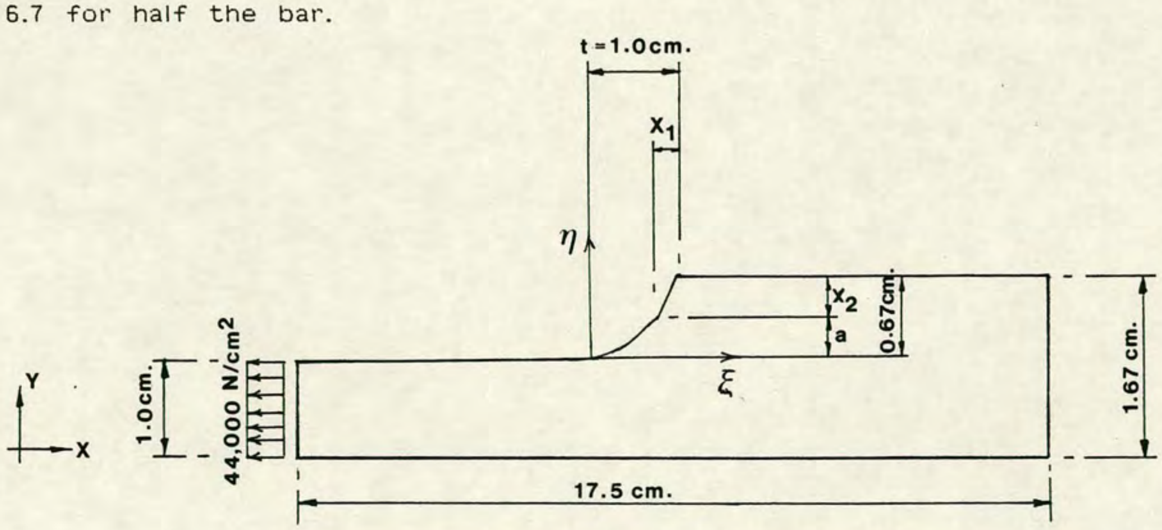


Figure 6.7 Fillet bar in tension

The local boundary to be varied is the transition curve, t. The bar was subjected to a tensile stress in the x-direction of 44,000 N/cm^2. Its elastic modulus was 2 X 10^7 N/cm^2, Poisson's ratio 0.25 and thickness of 0.1 cm. The dimensions of the problem are as shown in figure 6.7.

In reference [42], the transition curve was defined to be:

$$a\eta^2 + \xi^2 + 2b\eta = 0 \tag{6.3a}$$

where

$$a = \frac{t-x_1}{0.67-x_2}\left[\frac{t-x_1}{0.67-x_2} - \frac{2x_1}{x_2}\right] \tag{6.3b}$$

$$b = (t-x_1)\left[\frac{x_1}{x_2} - \frac{t-x_1}{0.67-x_2}\right] \quad \text{and} \tag{6.3c}$$

$$x_1 \geqslant 0 \ , \ x_2 \geqslant 0 \tag{6.3d}$$

where x_1 and x_2 are the coordinates of the tangent point to the curve. In reference [42] x_1 was set to zero for greater accuracy. The problem then becomes very simple as it only involves x_2. The trial and error procedure is now used to find the optimized shape as outlined in section 6.4.1.2.

The problem was first divided into 8-node isoparametric quadrilateral elements (QUAD8) as shown in figure 6.8(i). Only a fixed number of nodes were actually varied to find the new boundary using equation (6.3). The affected elements as a result of the node variation are shown hatched in figure 6.8(i) where unit load analyses are undertaken at these nodes in addition to those where there are applied loadings. Triangular elements are not useful in this particular example, as the severe distortion would give poor results.

6.4.1.2 Analysis of modified structures

To find the optimized shape, the trial and error procedure is outlined as follows. First the shapes for x_2=0.0,0.1,0.2.....0.6 are generated by using equation (6.3) with x_1=0.0. These generated shapes are shown in figures 6.8(i) to (vii).

For each shape, the theorems of geometric variation were used for the reanalysis to find its response. The stresses calculated at the Gauss points of the affected elements were extrapolated to the boundary nodes using the smoothing technique suggested by Hinton

FINITE ELEMENT MESH

NO. OF ELEMENTS = 33
NO. OF NODES = 128

Y-AXIS
X-AXIS

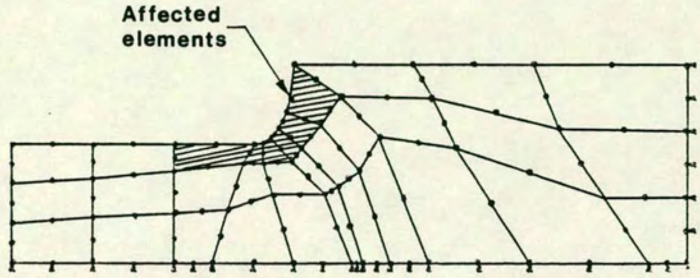


Figure 6.8(i) $X_2 = 0.0$

FINITE ELEMENT MESH

NO. OF ELEMENTS = 33
NO. OF NODES = 128

Y-AXIS
X-AXIS

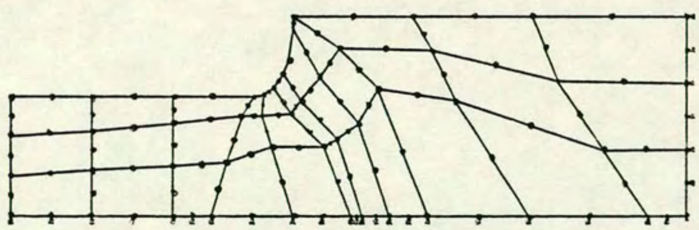


Figure 6.8(ii) $X_2 = 0.1$

FINITE ELEMENT MESH

NO. OF ELEMENTS = 33
NO. OF NODES = 128

Y-AXIS
X-AXIS

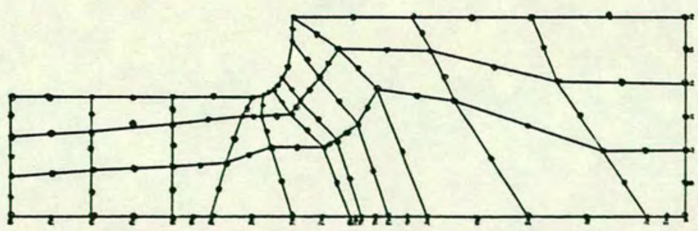


Figure 6.8(iii) $X_2 = 0.2$

FINITE ELEMENT MESH

NO. OF ELEMENTS = 33
NO. OF NODES = 128

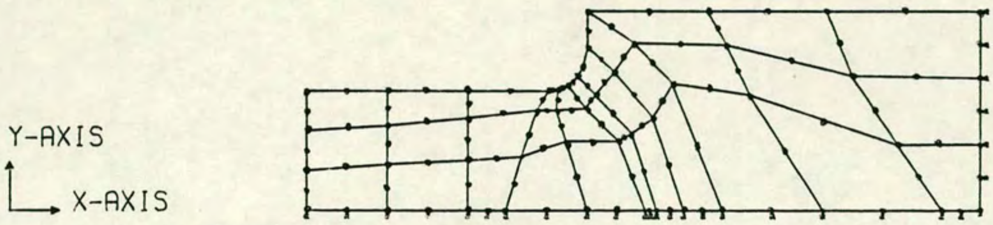


Figure 6.8(iv) $X_2 = 0.3$

FINITE ELEMENT MESH

NO. OF ELEMENTS = 33
NO. OF NODES = 128

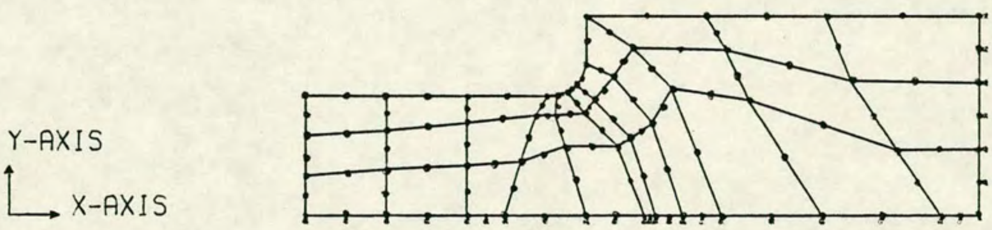


Figure 6.8(v) $X_2 = 0.4$

FINITE ELEMENT MESH

NO. OF ELEMENTS = 33
NO. OF NODES = 128

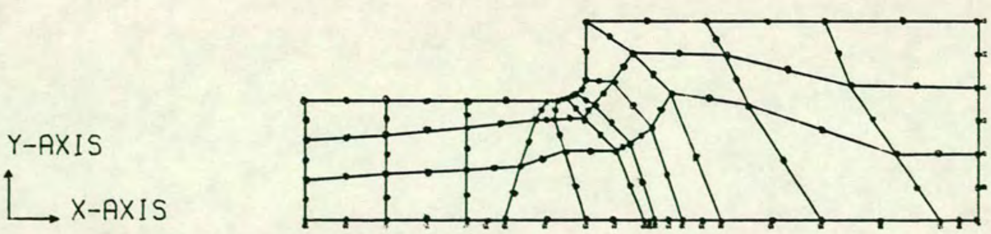


Figure 6.8(vi) $X_2 = 0.5$

FINITE ELEMENT MESH

NO. OF ELEMENTS = 33
NO. OF NODES = 128

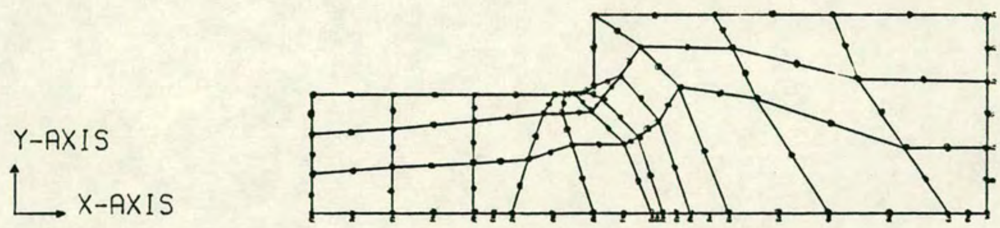


Figure 6.8(vii) $X_2 = 0.6$

FINITE ELEMENT STRUCTURE

PLOT OF PRINCIPAL STRESSES

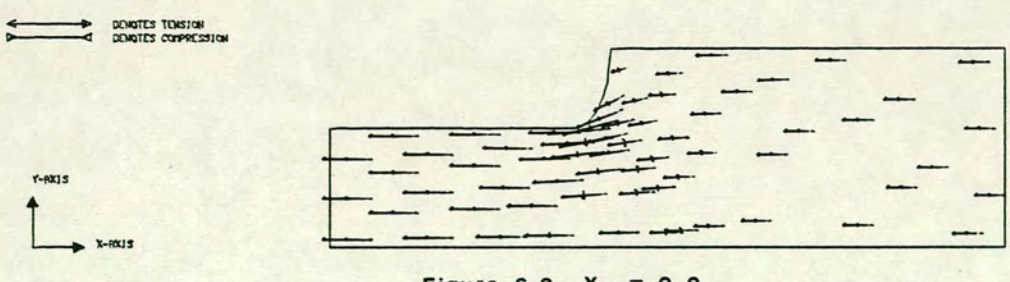


Figure 6.9 $X_2 = 0.0$

and Campbell[52]. The Gauss points are points within an element which are used in the numerical integration of the element stiffness matrix[154]. The stress concentration factor was then evaluated using equation (6.2). A plot of the principal stresses for the shape $x_2=0.0$ is shown in figure 6.9. Away from the transition curve the stress field is uniform as expected.

Figure 6.10(i) shows a plot of the maximum stress concentration factor against the variable x_2. The maximum stress concentration factor occurs at one of the boundary nodes that were being varied. The graph clearly indicates that an optimized shape exists for the smallest value of the maximum stress concentration factor. The next trial solution is to compute the shapes for the interval $0.30 \leqslant x_2 \leqslant 0.32$ obtained from the graph of figure 6.10(i). The values chosen were $x_2=0.3125$, 0.3150, 0.3175 and 0.3200. The results of the initial analysis were used again to find the response of the modified structures. A second plot in figure 6.10(ii), gives the optimized shape as $x_2=0.3176$ and $K_t=1.2754$. A further refinement is possible but not really necessary as the results are reasonably accurate.

A comparison of the CPU times taken for a fresh analysis and reanalysis by the theorems of geometric variation is tabulated in table 6.5. It is immediately obvious that reanalysis by the theorems is extremely efficient, although the time taken for the initial analysis of figure 6.8(i) is larger than that for a fresh analysis. However the subsequent reanalysis is only a fraction of a fresh or initial analysis and hence the total CPU time is less than for a series of fresh analyses. The CPU times for the subsequent reanalysis is the same because the number of varied nodes and affected elements remain the same. The reanalysis using the theorems is undertaken with reference to the original structural analysis.

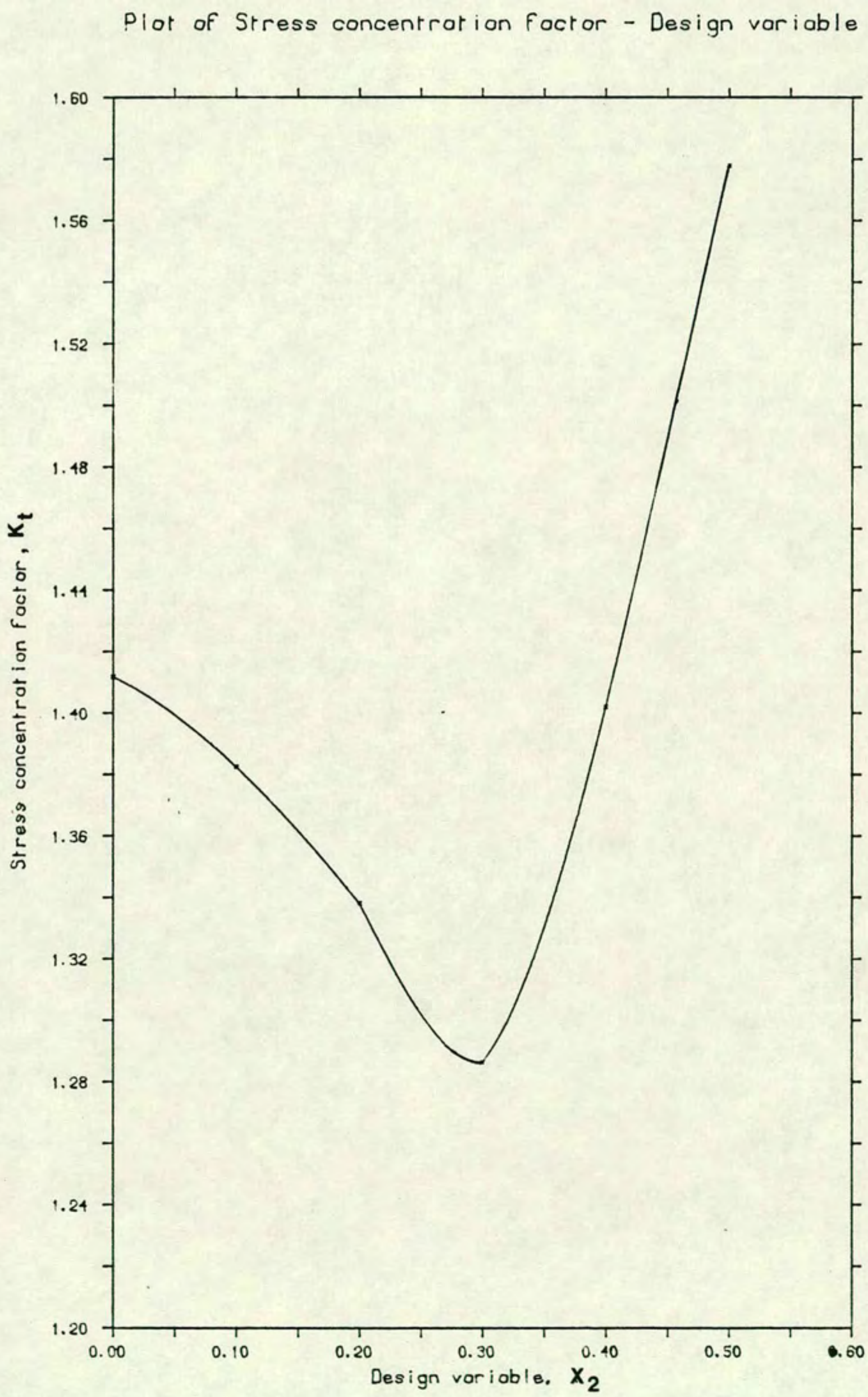

Plot of Stress concentration factor - Design variable
1.60
1.56
1.52
1.48
1.44
1.40
1.36
1.32
1.28
1.24
1.20
Stress concentration factor, K_t
0.00
0.10
0.20
0.30
0.40
0.50
0.60
Design variable, X_2

Figure 6.10(i)

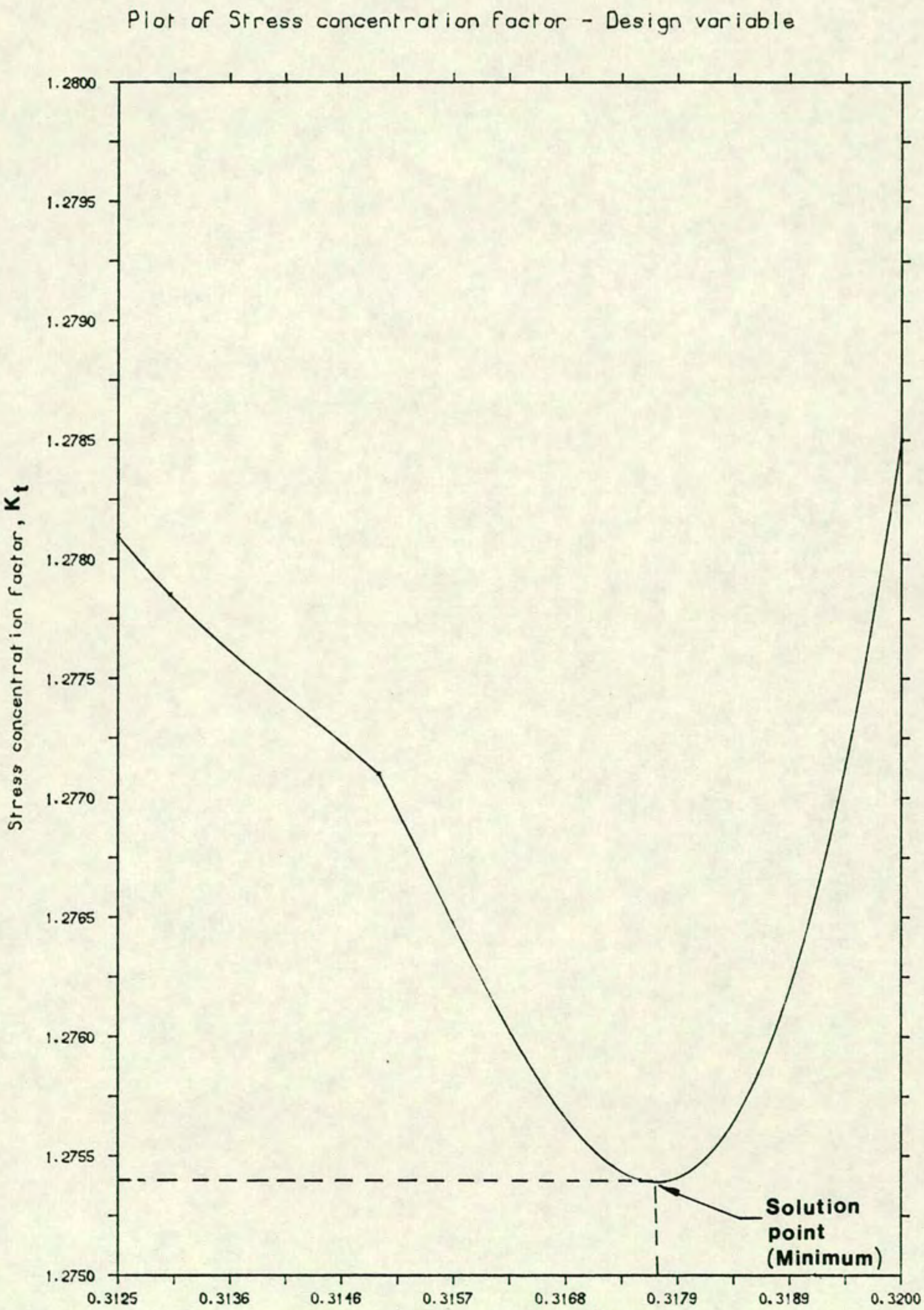


Figure 6.10(ii)

	CPU time taken (seconds)	
1st. trial and error (Figure)	Fresh analysis	Reanalysis by the theorems
x_2 =0.0 6.8(i)	5.27	9.55 (Initial analysis)
x_2 =0.1 6.8(ii)	3.59	1.84
x_2 =0.2 6.8(iii)	3.59	1.84
x_2 =0.3 6.8(iv)	3.59	1.84
x_2 =0.4 6.8(v)	3.59	1.84
x_2 =0.5 6.8(vi)	3.59	1.84
x_2 =0.6 6.8(vii)	3.59	1.84
2nd. trial and error (No figures)		
x_2 =0.3125	3.59	1.84
x_2 =0.3150	3.59	1.84
x_2 =0.3175	3.59	1.84
x_2 =0.3200	3.59	1.84
Total CPU time Σ	41.17	27.95

Table 6.5 CPU time for reanalysis

If a large number of reanalysis is required it is best to use the theorems of geometric variation. This might occur in optimization design for example where many analyses for modified structures are required. For a small number of reanalysis, a fresh analysis is more efficient. For example, if only two reanalysis is required then two fresh analyses take only 8.86 seconds compared to 11.39 seconds using the theorems. Therefore the theorems would be efficient for locally modified structures requiring a large number of reanalysis.

6.4.2 Unconfined seepage flow analysis

The flow of water through a porous medium for a free surface is of practical importance to engineers. There are various techniques of solution some of which include the sketching of flow nets and the finite difference technique. More recently the finite element method has proven to be very effective. Most of the finite element procedures for free surface analysis involve successive modifications of the mesh[43] and this is the procedure that would be amenable for the theorems of geometric variation.

So far in this thesis, the discussions have been centred on the analysis of structures. The seepage flow analysis is a field problem with only one unknown, the head or potential. A finite element program for structural analysis requires little modifications to deal with such problems. The following is a brief outline of the problem and the analogies with a structural problem is discussed. Figure 6.11 shows a region of interest with the appropriate boundary conditions.

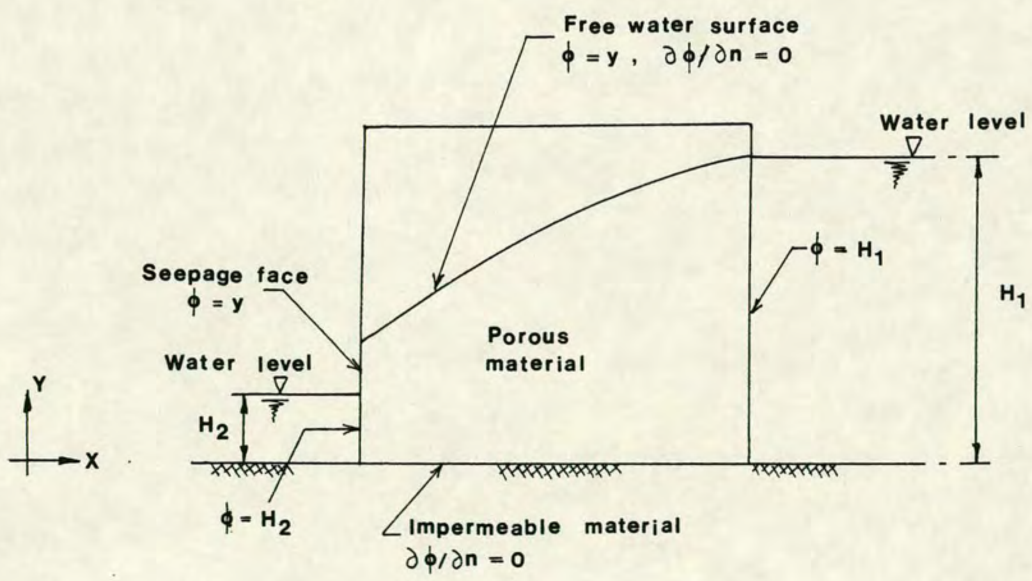


Figure 6.11 Unconfined seepage flow

The unknowns of the problem is the head or potential, ϕ. The outward normal to a boundary is n, and $\partial\phi/\partial n$ is the gradient of the head. The analogies with a structural problem are:

Seepage flow	Structural problem
Φ - head or potential	$\{\delta\}$ - displacements
[k] - permeability matrix	[D] - elasticity matrix
Q - discharge	{F} - applied loads
{v} - velocity vector	$\{\sigma\}$ - stress vector

The specified boundary conditions in figure 6.11 are equivalent to the prescribed displacements in a structural problem. An important difference is that a field problem is a one degree of freedom problem. Therefore at a node of a finite element mesh there will only be one unknown in this case the head. Otherwise the finite element method is the same as given in Chapter 2.

There are two boundary conditions for the free surface which cannot be satisfied simultaneously. These are when $\Phi=y$ where y is the elevation head and $\partial\Phi/\partial n=0$ which indicates that there is no flow perpendicular to the free surface. The usual procedure is to guess the free surface first and then analyse the problem by the finite element method. If the Φ's of the free surface nodes are not equal to the elevation head, y the analysis must be repeated with a new mesh. Instead of updating the whole mesh only the free surface nodes are updated. This is undertaken by allowing the y-coordinates of the free surface nodes to be equal to the Φ's calculated previously. After repeated iterations the values of the Φ's and y's of the free surface nodes should be the same or within some specified tolerance. In most analyses only the stiffness of the elements adjacent to the free surface are evaluated throughout the iterations.

6.4.2.1 Initial analysis

The problem shown in figure 6.13 was first studied by France et.al.[43] and subsequently by Bathe and Khoshgoftaar[15]. In the figure the dimensions and material properties of the problem are given in imperial units, where k_x and k_y are the permeabilities in the x and y directions respectively. The objective is to find the free water surface through the porous block medium as the water flows from an

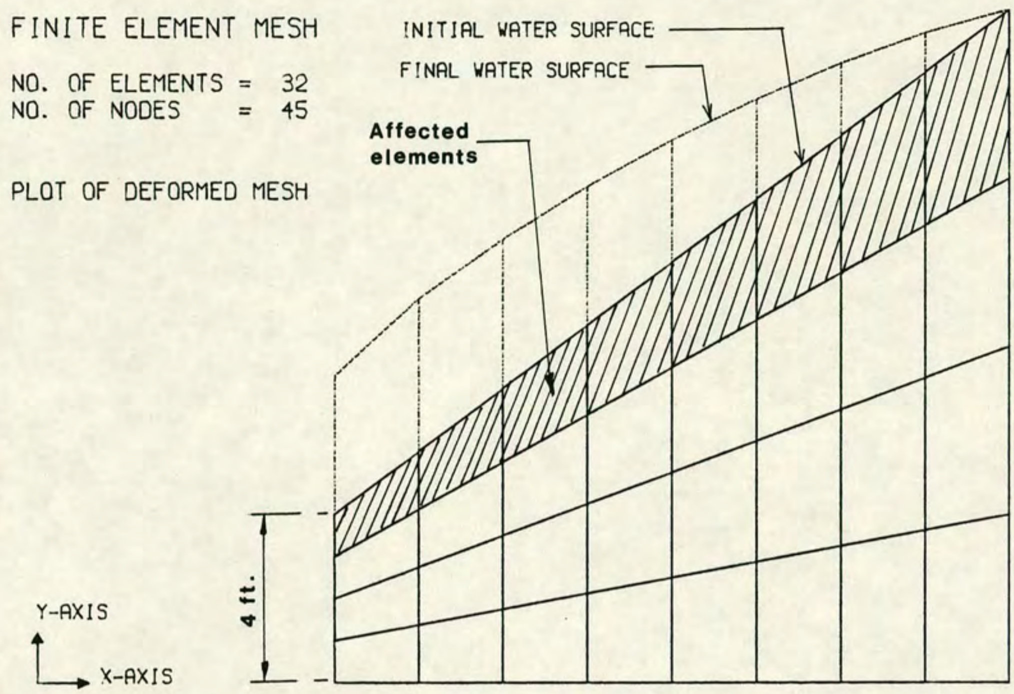


Figure 6.12(i) Coarse mesh

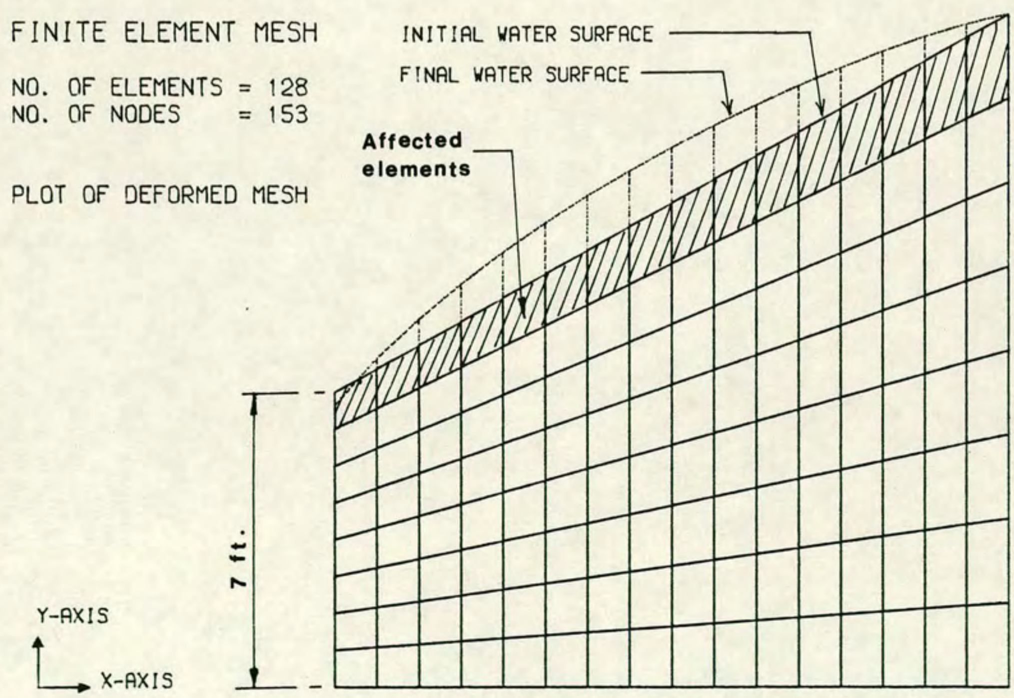


Figure 6.12(ii) Fine mesh

elevation of 16 ft. to 0 ft. The initial guess of the free surface is shown in figure 6.12(i) for a coarse mesh of QUAD4 elements. This guess is obviously incorrect, as the water surface cannot simply be a straight line between two different elevations. The seepage face is 4 ft. above the lower water level. Unit load analyses (or rather unit discharges) are carried out at the nodes of the affected elements shown hatched in figure 6.12(i).

6.4.2.2 Analysis of modified meshes

The mesh is modified according to that outline in section 6.4.2. The theorems of geometric variation are then used to calculate the new heads for the modified mesh. The process converges after 16 iterations. A plot of the deformed mesh at the final iteration is shown in figure 6.12(i). The deformed mesh is the shape of the free surface of the water flowing through the porous medium.

As a further a refinement a finer mesh of QUAD4 elements was used as shown in figure 6.12(ii). The results of the coarse mesh were used to estimate the height of the seepage face. This was set to 7 ft. with the initial free surface as shown in figure 6.12(ii). The theorems were then used for this mesh, and the final free surface was obtained after 30 iterations as shown in figure 6.12(ii).

A graph of the height of the free surface against the distance from the seepage face was plotted in figure 6.13. The results of the coarse and fine mesh are very near to the results of France's et.al.[43]. The differences may be attributed to using different elements and meshes.

The comparison for the CPU times taken for fresh analysis and reanalysis by the theorems are tabulated in table 6.6. The number of iterations is equal to the number of fresh analysis or reanalysis.

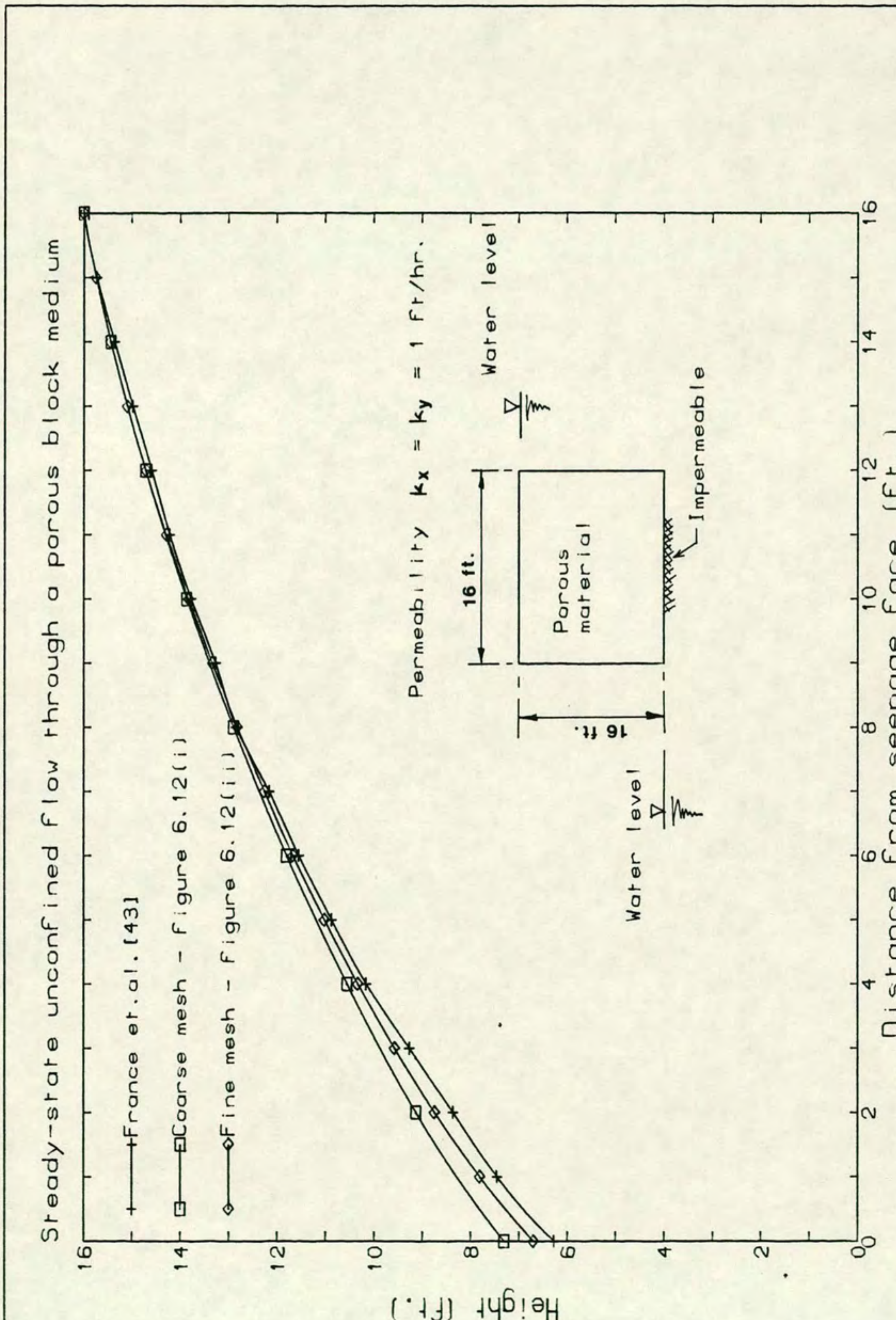

Steady-state unconfined flow through a porous block medium
France et.al. [43]
Coarse mesh - Figure 6.12(i)
Fine mesh - Figure 6.12(ii)
Permeability $k_x = k_y = 1$ ft/hr.
16 ft.
Porous material
Impermeable
Water level
Water level
16 ft.
Height (ft.)
Distance from seepage face (ft.)
0
2
4
6
8
10
12
14
16

Figure 6.13

	Total CPU time taken (seconds)	
Type of mesh (Figure)	Fresh analysis	Reanalysis by the theorems
Coarse mesh 6.12(i) Analysis for 16 iterations	3.17	2.84
Fine mesh 6.12(ii) Analysis for 30 iterations	20.20	14.14

Table 6.6 CPU time for reanalysis

For a large number of reanalysis the theorems are again shown to be efficient. In addition, as the structure becomes large (a finer mesh in this case) the time savings begin to be substantial as indicated in table 6.6. The reanalysis is always referred to the original structural analysis.

6.5 Summary

The use of the theorems of geometric variation as an efficient reanalysis technique have been illustrated using the formulations of Chapter 3. The proposed technique results in significant savings in cost and time. It will be particularly effective for locally modified large structures with a large semi-bandwidth.

The theorems of geometric variation are also more versatile than the theorems of structural variation. If material changes are also considered they may easily be accommodated at the same time as suggested in Chapter 3. Various combinations of applied loadings may also be considered provided that they act at the same nodes as in the initial analysis. However it should be noted that the use of triangular elements with node variations is not recommended. This is because of the severe distortions of the elements as was shown in section 6.3. The 8-node isoparametric quadrilateral elements is a superior element to use since it is more accurate.

CHAPTER 7

EFFICIENCY OF THE THEOREMS OF GEOMETRIC VARIATION FOR MATERIAL NONLINEAR PROBLEMS

7.1 Introduction

In this chapter, the theorems of geometric variation are developed for the reanalysis of material nonlinear problems. The matrix forms of the proposed techniques that have been derived in Chapter 4 are investigated for efficiency. The comparisons are based on the CPU time taken for the analysis in comparison with that of the conventional Newton-Raphson methods.

Four benchmark examples were used to study the efficiency of the proposed techniques. These examples will be used to select which of these techniques are the most efficient as well as selecting the type of problems that they may be used effectively.

7.2 Material nonlinear analysis by the theorems of geometric variation

The material nonlinear problems that were studied using the reanalysis techniques were restricted to the class of elasto-plastic materials which were briefly discussed in Chapter 2. The elasto-plastic matrix for various materials may be found in the texts of Owen and Hinton[112] and Zienkiewicz[154]. The examples here only consider Von Mises material with isotropic strain-hardening properties.

The proposed nonlinear reanalysis techniques do not depend on how the elasto-plastic matrix is formed, they are concerned with the solution of the nonlinear equilibrium equations. This is undertaken using a sequence of linear analysis which has been proved to be efficient in Chapter 6. The most important part of the analysis is undoubtly in the evaluation of the compensation forces which is common to all the proposed techniques. This evaluation is given by equation (4.13) of Chapter 4. The analysis is now briefly outlined as follows.

7.2.1 Initial analysis

The elasto-plastic analysis of a structure by the theorems of geometric variation first requires an assumption regarding which part of the structure is likely to become plastic. These are shown hatched in the figures illustrating the examples of section 7.3. Unit load analyses are performed at the nodes of the assumed plastic elements in addition to those nodes where there are applied loadings. These nodes are the 'a' affected nodes and 'u' unaffected nodes respectively. The unit load analyses are given by:

$$[K_o][\delta_a \mid \delta_u] = [I_a \mid I_u] \tag{7.1}$$

The assumption concerning which part of the structure is likely to be plastic requires experience and careful physical evaluation. This assumption is very much the same as that required in the substructure analysis of elasto-plastic problems[122], where the assumed elastic and plastic parts are divided into substructures.

7.2.2 Analysis of modified structure

After the initial analysis has been performed, the response of the modified structure may now be obtained. This response is given by equations (4.12) and (4.15) depending on the technique chosen in the analysis. For the initial stiffness technique, equation (4.12) is used and the compensation forces need not be calculated and the scale factors are directly given by the residual forces.

Other proposed techniques require the evaluation of the compensation forces. If an element has become plastic, then its modified or tangential stiffness is given by:

$$[K_T^e] = \int_{\Omega e} [B]^T [D_{ep}][B] \, d\Omega_e \tag{7.2}$$

Equation (7.2) represents a reduction of the element stiffness when plasticity effects are taken into account. Hence the change in element stiffness is:

$$\begin{aligned} [\Delta K^e] &= [K_T^e] - [K_o] \\ &= \int_{\Omega e} [B]^T [D_{ep}][B] \, d\Omega_e - \int_{\Omega e} [B]^T [D][B] \, d\Omega_e \end{aligned}$$

$$= \int_{\Omega_e} [B]^T ([D_{ep}]-[D])[B] \; d\Omega_e \tag{7.3}$$

However it was shown in equation (2.20a) that the $[D_{ep}]$ matrix may be decomposed into an elastic and plastic parts:

$$[D_{ep}] = [D] - [D_p] \tag{7.4}$$

Substitution of equation (7.4) into (7.3) gives:

$$[\Delta K^e] = - \int_{\Omega_e} [B]^T [D_p][B] \; d\Omega_e \tag{7.5}$$

Equation (7.5) may now be used to form the matrix of compensation forces as given by equation (4.13). For an elasto-plastic material this is:

$$[C^e_{aa} \mid C^e_{au}] = - \int_{\Omega_e} [B]^T [D_p][B] \; d\Omega_e \quad [\delta^e_a \mid \delta^e_u] \tag{7.6}$$

The overall matrix of compensation forces are then formed by adding the contribution of $[C^e_{aa} \mid C^e_{au}]$ for each plastic element. The scale factors may be determined using equation (4.14) or (4.15c) and hence the new displacements may be calculated using equation (4.15a). Depending on the technique selected, the matrix of compensation forces may be reformed during each iteration or at some chosen intervals as discussed in Chapter 4.

7.3 Investigation of the efficiency

Four benchmark examples were considered to assess the efficiency of the proposed techniques of Chapter 4. The examples are the elasto-plastic analysis of a plain strain thick cylinder, a simply supported circular plate, a perforated plate and a notched beam.

The CPU times for each of the proposed techniques were compared with each other. Other comparisons were undertaken with the usual procedure of Newton-Raphson and its degenerate forms as discussed in Chapter 2. A comparison of CPU times is important to judge whether the proposed techniques are suitable. In Chapter 6, the theorems of geometric variation were shown to be efficient for a series of linear elastic reanalyses. Therefore it would be expected that the proposed techniques are efficient for nonlinear analysis. This is because the nonlinear analysis is performed by a sequence of linear analysis. However the degree of efficiency of the proposed techniques is unknown. This is investigated in the following problems to assess certain characteristics of the proposed techniques.

7.3.1 Plain strain thick cylinder

The finite element analysis of this cylinder was obtained from reference [112]. Three different meshes were used as shown in figures 7.1(i), (ii) and (iii) for a quarter of the cylinder. Figures 7.1(i) and (ii) are for a coarse and fine mesh of QUAD8 elements respectively. Figure 7.1(iii) is a fine mesh of QUAD4 elements. The cylinder was subjected to an internal pressure, P which was incremented by a load factor, λ. A total of six load increments were used for λ=0.4, 0.5, 0.6, 0.7, 0.8 to 0.9. Plain strain conditions were assumed and the cylinder was loaded until collapse. The material properties, dimensions and boundary conditions are shown in figure 7.1(i). The semi-bandwidth, B of each mesh is also shown in the figures.

To use the theorems of geometric variation, an initial load analysis is required. The area in which the plasticity is assumed to be confined is shown in figure 7.1(ii). This area may be arrived at by physical considerations. Plasticity is initiated from the inner radius to the outer one as the load is incremented. Since, it was loaded up to collapse, the area includes a large number of elements likely to be plastic which is reasonable.

Figure 7.1(iii) shows which elements have become plastic as λ increases. This is not a contour plot of plasticity. Plasticity spreads from the inner to the outer radius as expected. The collapse load was obtained at λ=0.9 or at a pressure of P=180N/mm^2. A plot of the load factor against the displacement at node 1 is given in figure 7.2. Below λ=0.5 the cylinder is still elastic and beyond λ=0.9, the solution diverges indicating collapse. The results of the nonlinear analysis using the theorems of geometric variation were identical to that of the Newton-Raphson methods.

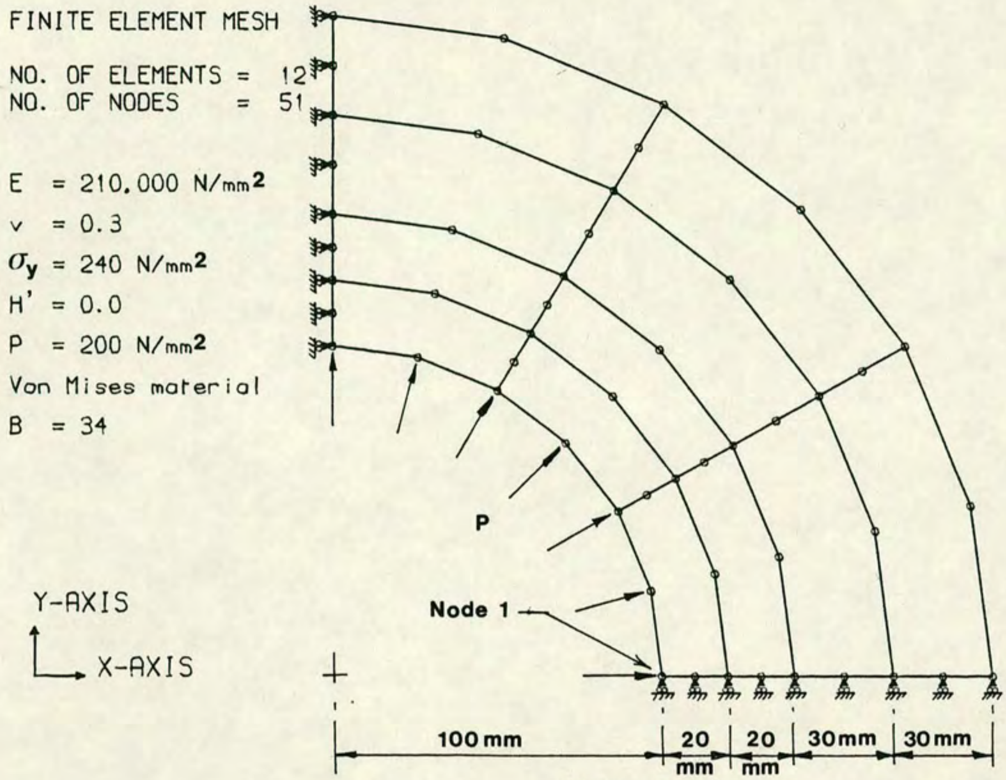


Figure 7.1(i)

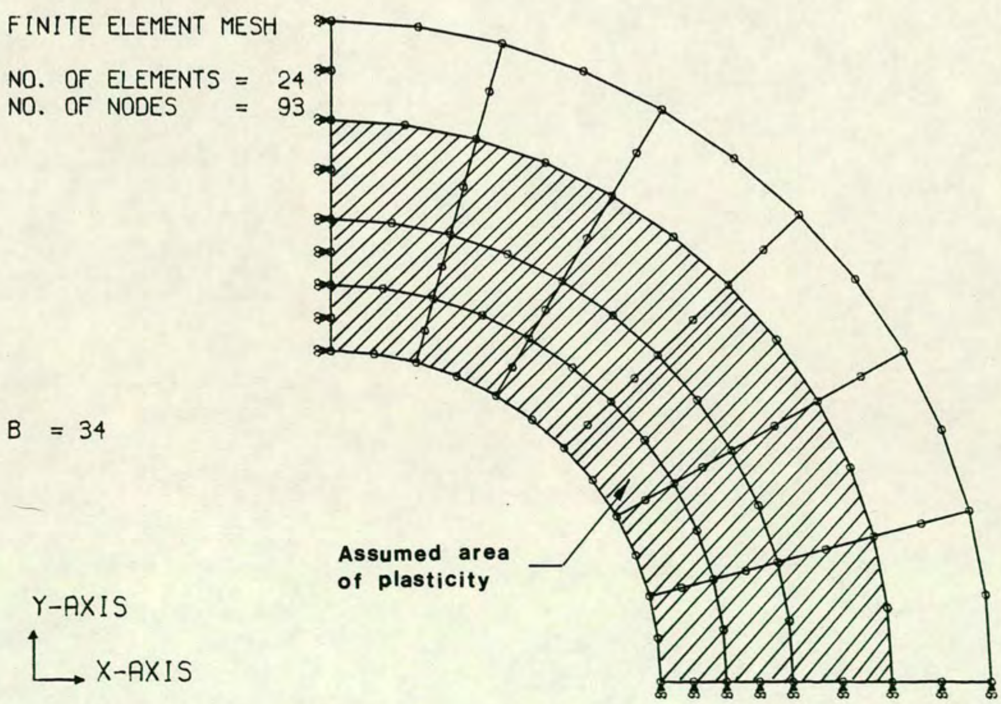


Figure 7.1(ii)

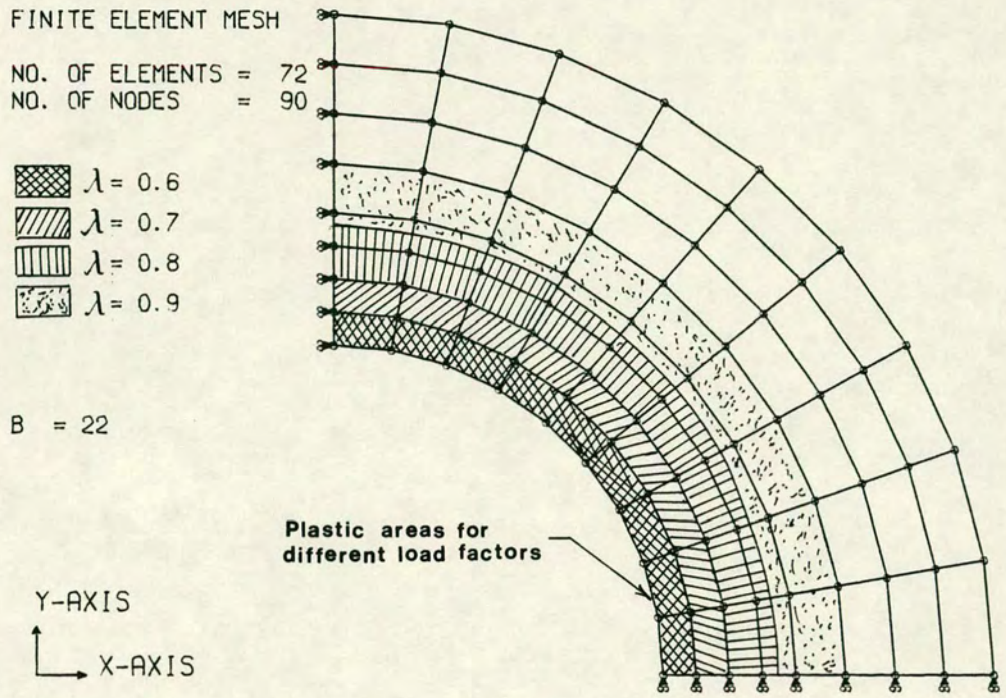


Figure 7.1(iii)

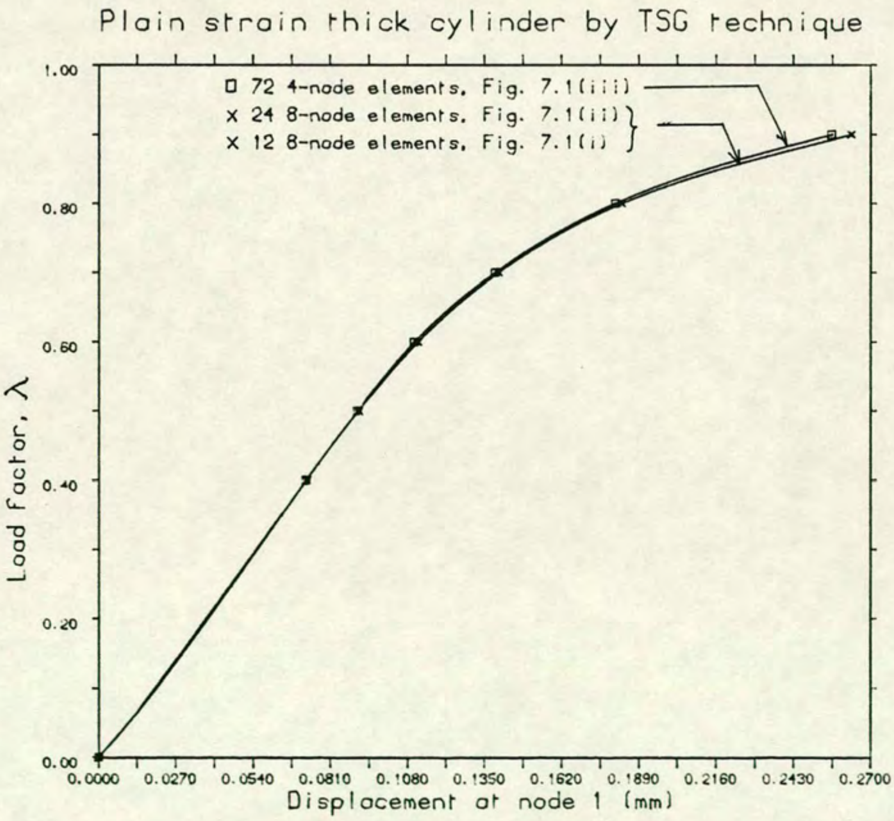


Figure 7.2

7.3.1.1 Graphs and tables of CPU time

Figures 7.3(i),(ii),(iii), 7.4(i),(ii),(iii) and 7.5(i),(ii),(iii) are graphs of the load factor, λ against the CPU time taken for the analyses. The abbreviations in each figure require further explanation as follows (for further reference a list of the nonlinear analysis techniques used in this thesis may be found in the section on notations).

- **Figures 7.3(i),7.4(i) and 7.5(i)**

- **IS technique** : The initial stiffness technique by the Newton-Raphson method as described in section 2.4.1.2. This technique is used for comparison with the ISG and ISVG techniques.

- **ISG technique** : The initial stiffness technique by the theorems of geometric variation as described in section 4.2.2.1.

- **ISVG technique** : The variant form of the initial stiffness technique by the theorems of geometric variation as described in section 4.2.2.4.

- **Figures 7.3(ii),7.4(ii) and 7.5(ii)**

- **TS technique** : The tangential stiffness technique by the Newton-Raphson method as described in section 2.4.1.1. This technique is used for comparison with the TSG technique.

- **TSV technique** : The tangential stiffness technique by the Newton-Raphson method. Here the elastic stiffness is used at the first iteration of every load increment. This is for comparison with the TSVG technique.

- **TSG technique** : The tangential stiffness technique by the theorems of geometric variation as described in section

4.2.2.2.

- **TSVG technique** : The variant form of the tangential stiffness technique by the theorems of geometric variation as described in section 4.2.2.4.

- **Figures 7.3(iii),7.4(iii) and 7.5(iii)**

- **ITS technique** : A combination of the initial and tangential stiffness technique by the Newton-Raphson method. This is described in section 2.4.1.3 and used for comparison with the ITSG technique.

- **ITSV technique** : This technique is similar to the ITS technique. At the first iteration of every load increment the elastic stiffness is used. It is only updated at the second iteration and kept constant until the next load increment. It is used for comparison with the ITSVG technique.

- **ITSG technique** : The initial/tangential stiffness technique by the theorems of geometric variation as described in section 4.2.2.3.

- **ITSVG technique** : The variant form of the initial/tangential stiffness technique by the theorems of geometric variation as described in section 4.2.2.4.

Tables 7.1(i),(ii) and (iii) show the total CPU time taken for each analysis by one of the proposed techniques for the three finite element idealisations of figures 7.1(i),(ii) and (iii). In the tables, the number of iterations required is the total number of fresh analysis or reanalysis. It is sum of the number of iterations for each load increment.

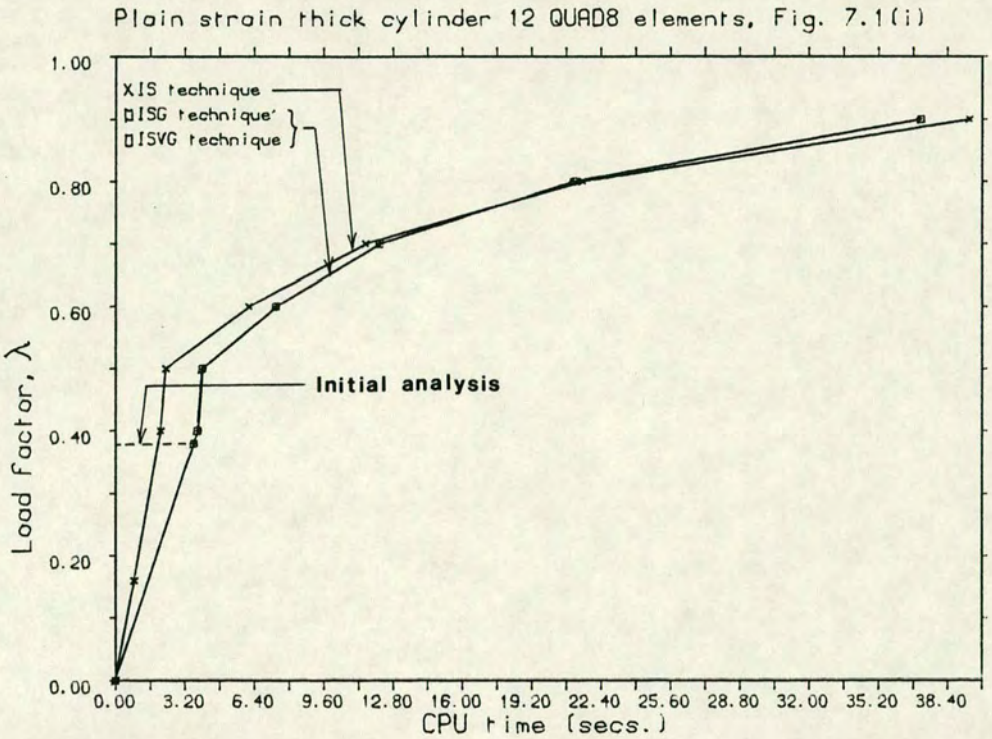
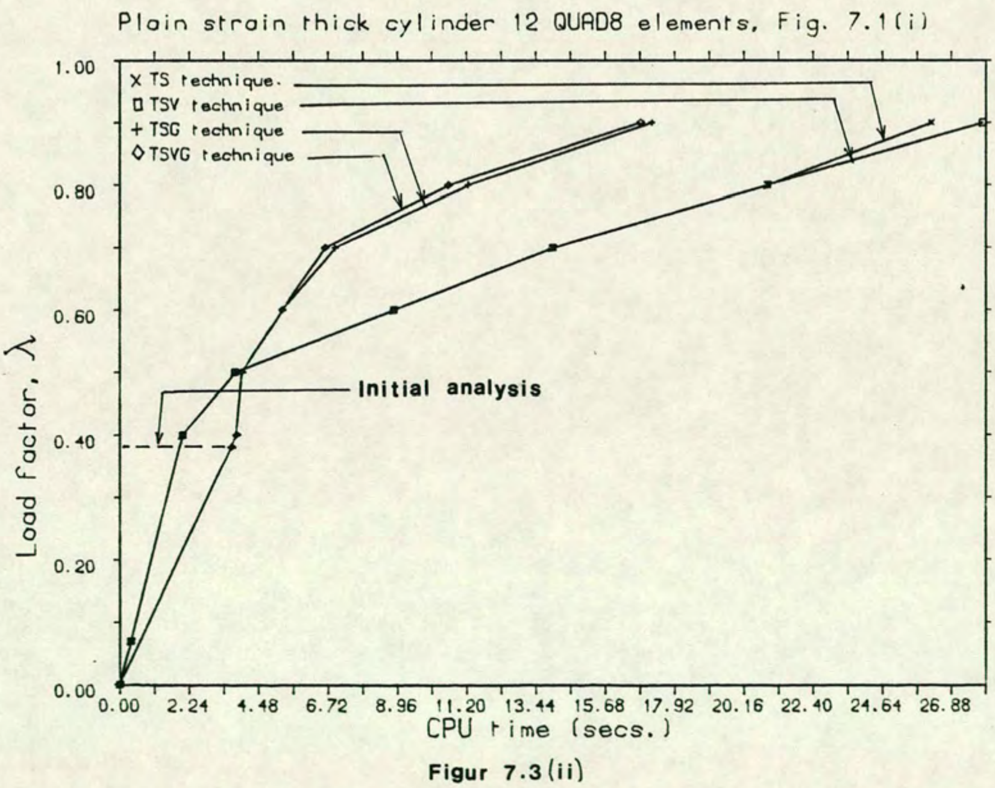


Figure 7.3(i)



Figur 7.3(ii)

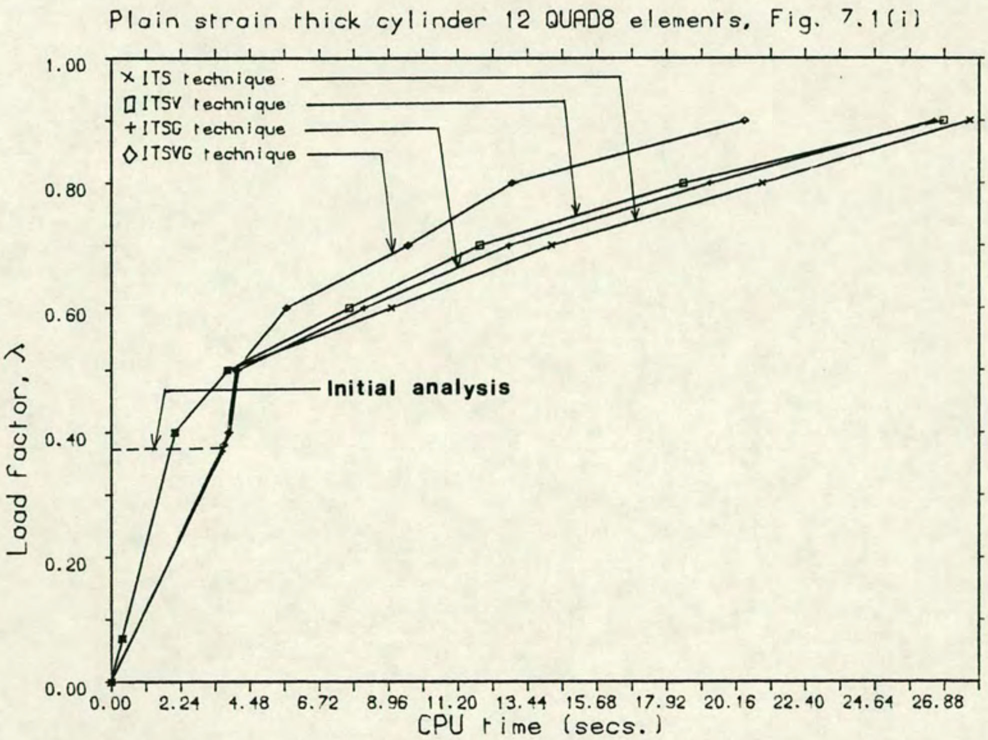


Figure 7.3(iii)

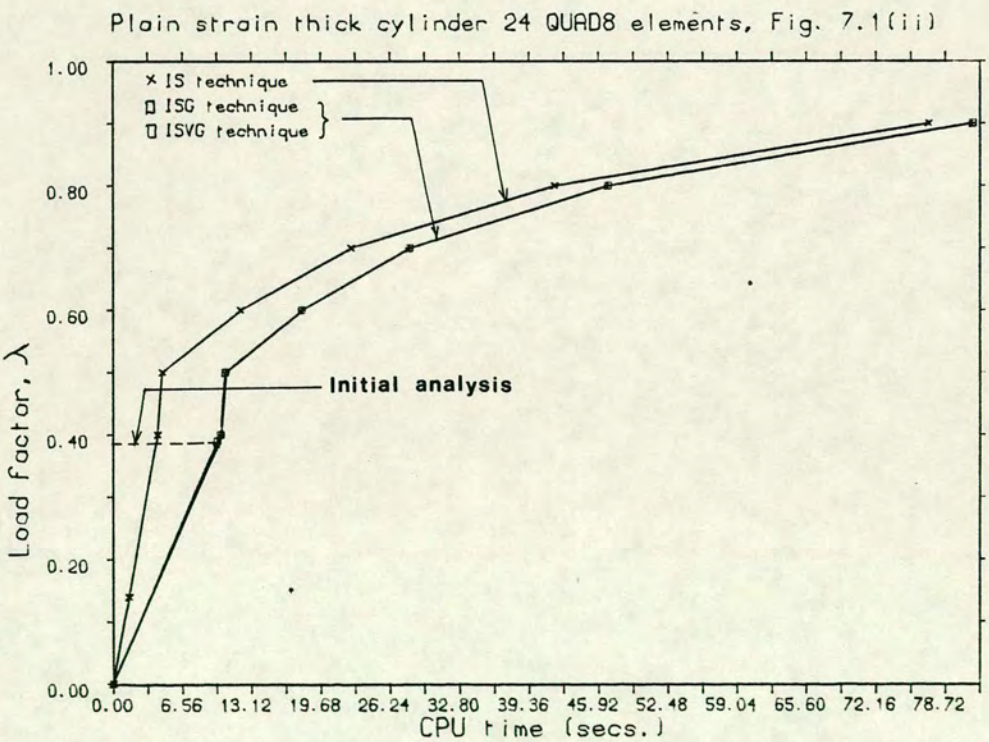


Figure 7.4(i)

Plain strain thick cylinder 24 QUAD8 elements, Fig. 7.1(ii)

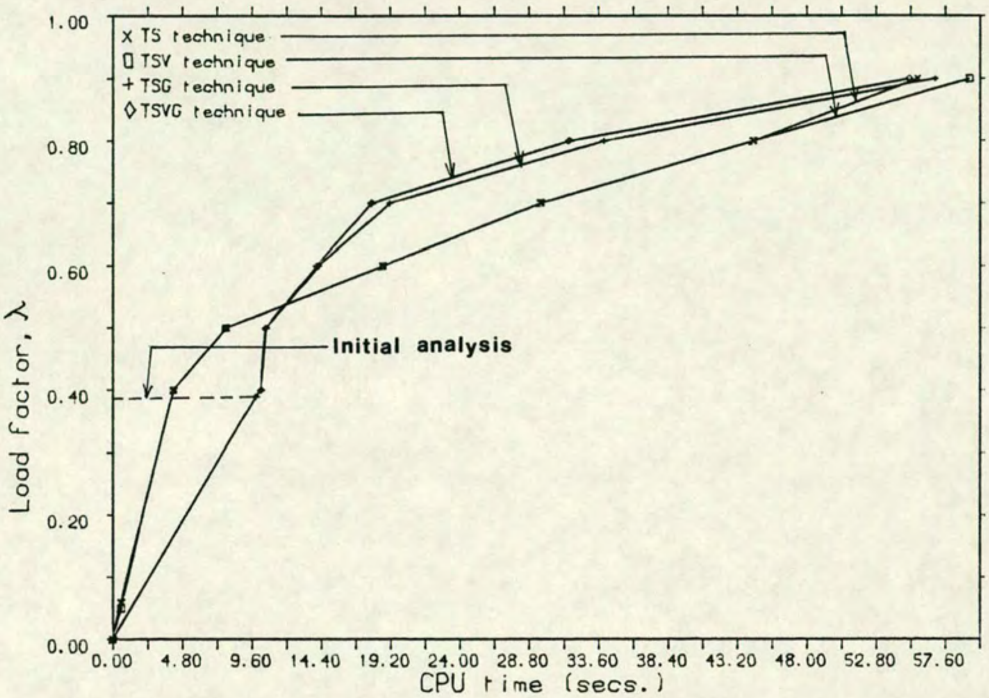


Figure 7.4(ii)

Plain strain thick cylinder 24 QUAD8 elements, Fig. 7.1(ii)

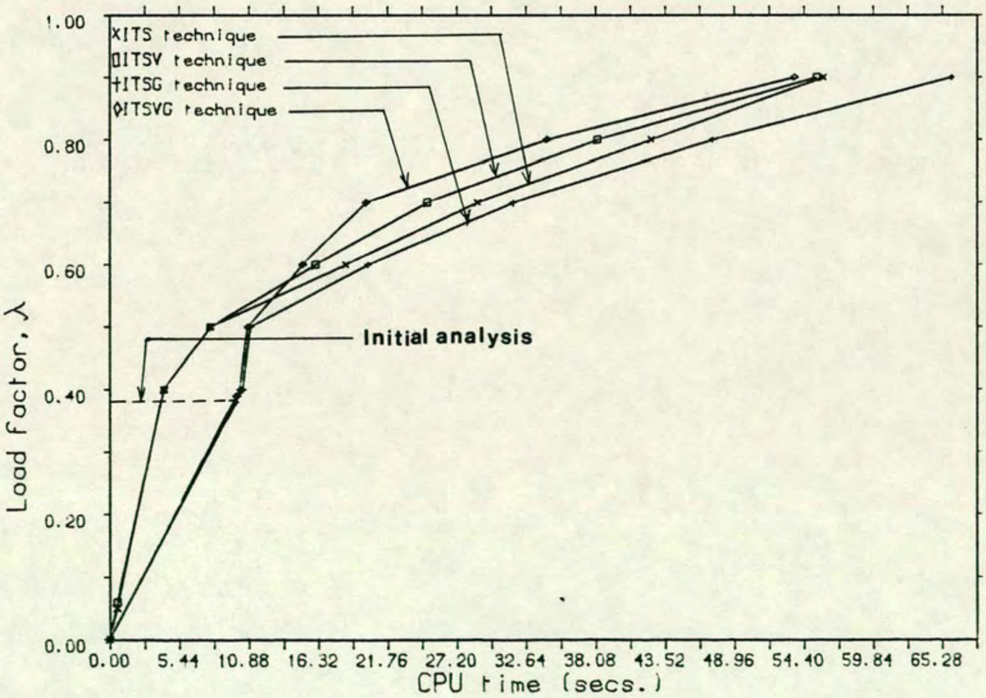


Figure 7.4(iii)

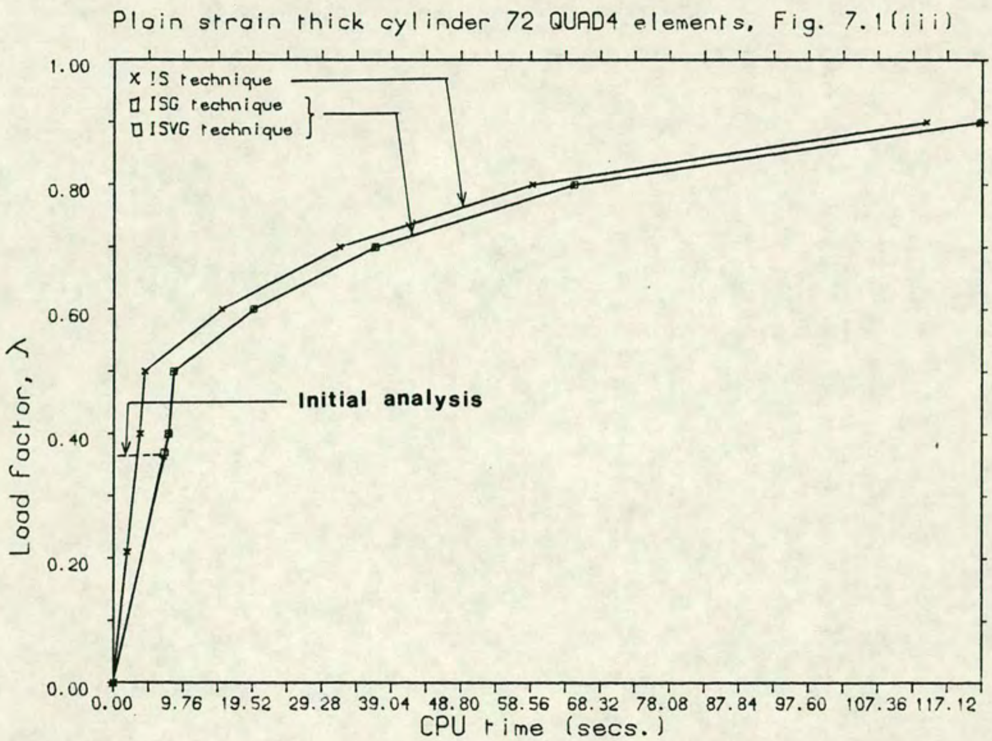


Figure 7.5(i)

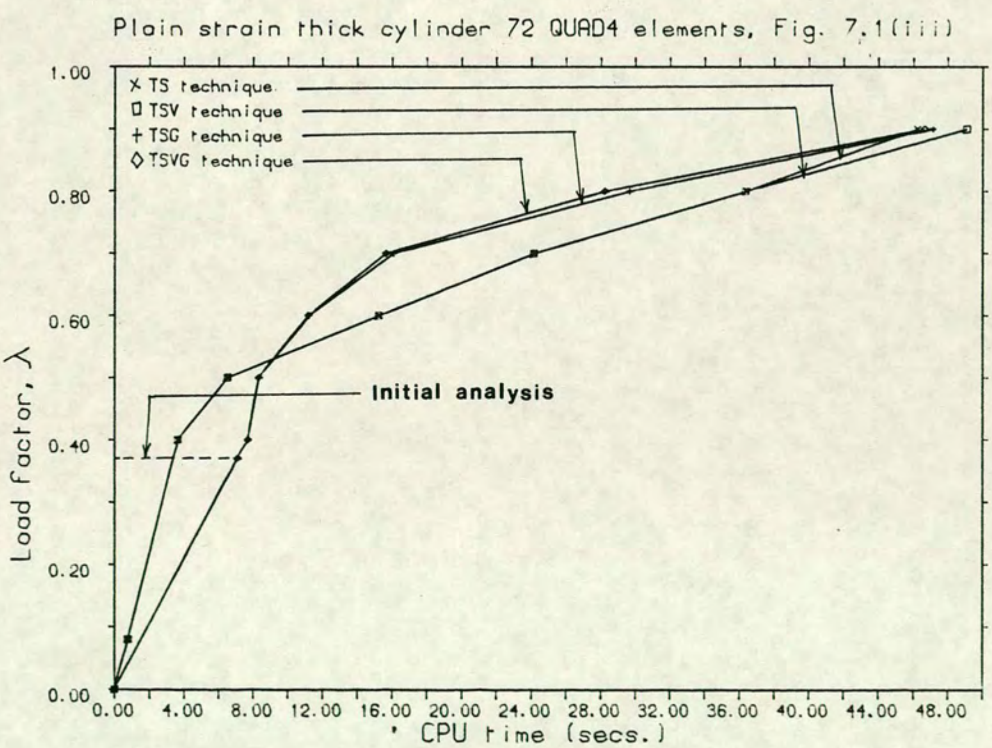


Figure 7.5(ii)

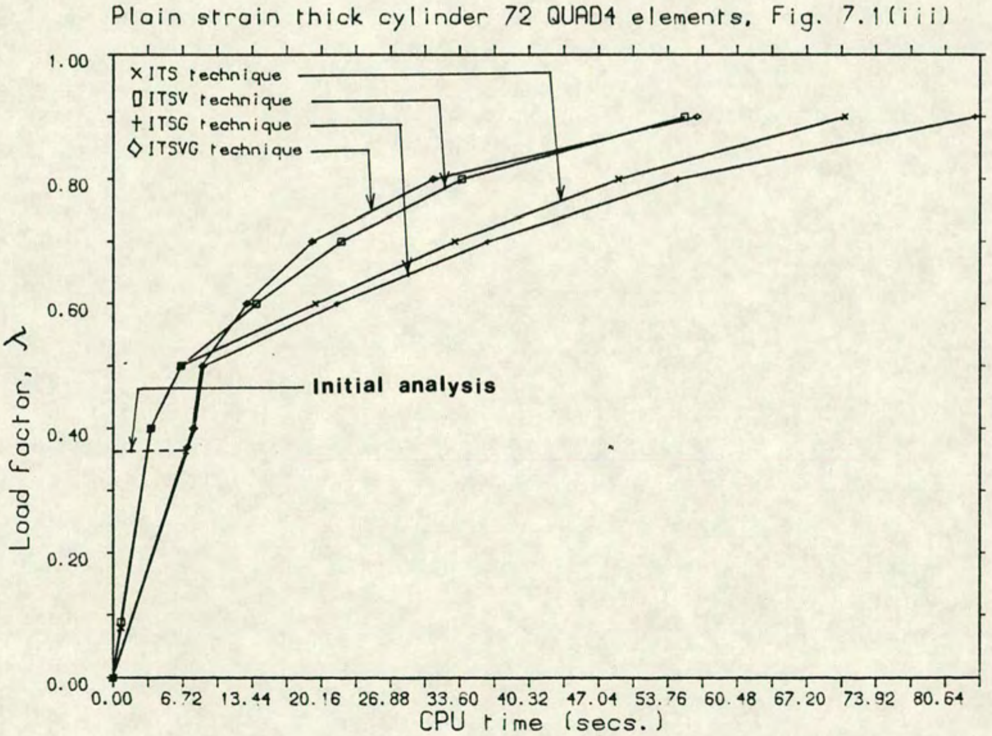


Figure 7.5(iii)

Plain strain thick cylinder 12 QUAD8 elements Figure 7.1(i)			
Technique	Graph (Figure)	No. of iterations required	Total CPU time (secs.) for λ=0.0 to 0.9
IS	7.3(i)	150	39.40
ISG	7.3(i)	150	37.07
ISVG	7.3(i)	150	37.14
TS	7.3(ii)	15	26.23
TSV	7.3(ii)	16	27.91
TSG	7.3(ii)	15	17.21
TSVG	7.3(ii)	16	16.80
ITS	7.3(iii)	74	27.70
ITSV	7.3(iii)	75	26.87
ITSG	7.3(iii)	74	26.54
ITSVG	7.3(iii)	75	20.42

Table 7.1(i) Total CPU time taken

Plain strain thick cylinder 24 QUAD8 elements Figure 7.1(ii)			
Technique	Graph (Figure)	No. of iterations required	Total CPU time (secs.) for λ=0.0 to 0.9
IS	7.4(i)	149	77.13
ISG	7.4(i)	149	81.20
ISVG	7.4(i)	149	81.32
TS	7.4(ii)	15	55.68
TSV	7.4(ii)	16	59.28
TSG	7.4(ii)	15	56.93
TSVG	7.4(ii)	16	55.07
ITS	7.4(iii)	74	55.90
ITSV	7.4(iii)	75	55.39
ITSG	7.4(iii)	74	66.02
ITSVG	7.4(iii)	75	53.65

Table 7.1(ii) Total CPU time taken

Plain strain thick cylinder 72 QUAD4 elements Figure 7.1(iii)			
Technique	Graph (Figure)	No. of iterations required	Total CPU time (secs.) for λ=0.0 to 0.9
IS	7.5(i)	139	114.16
ISG	7.5(i)	139	121.58
ISVG	7.5(i)	139	121.71
TS	7.5(ii)	15	46.20
TSV	7.5(ii)	16	49.08
TSG	7.5(ii)	15	47.15
TSVG	7.5(ii)	16	46.62
ITS	7.5(iii)	72	70.86
ITSV	7.5(iii)	40	55.41
ITSG	7.5(iii)	72	83.50
ITSVG	7.5(iii)	40	56.59

Table 7.1(iii) Total CPU time taken

7.3.1.2 Discussion of results

From tables 7.1(i),(ii) and (iii), the theorems are not efficient when the structure is loaded up to collapse, except when the structure is small as in the case of the idealisation of figure 7.1(i). This is because the $[C_{aa}]$ matrix becomes large in comparison to the tangential stiffness matrix of the structure when there are many plastic elements. The $[C_{aa}]$ matrix is full and unsymmetric and hence requires more operations to reduce it than the tangential stiffness matrix.

However the tables do not quite reveal the whole story. The graphs of figures 7.3,7.4 and 7.5 give the CPU time taken during the loading history of the structure. At low load factors of λ=0.4 to

0.5, the theorems are inefficient because the initial analysis required offsets the efficiency in the reduction of a small $[C_{aa}]$. As the load factor increases to about $\lambda=0.6$ to 0.8, the theorems begin to be efficient except for the ISG and ISVG techniques. This is particularly true for the TSG and TSVG techniques shown in figures 7.3(ii),7.4(ii) and 7.5(ii). These two techniques are in fact the most efficient in comparison to the other proposed techniques.

The ISG and ISVG techniques do not seem to have any advantages to that of the usual procedures as may be seen from figures 7.3(i),7.4(i) and 7.5(i). The ITSG and ITSVG techniques combine both the advantages and disadvantages of the initial and tangential stiffness techniques.

7.3.2 Simply supported circular plate

This problem was analysed in references [112,140] and treated as an axisymmetric problem. The finite element mesh for half the plate, dimensions, material properties and boundary conditions are as shown in figure 7.6(i).

For the initial analysis, the area in which plasticity was assumed to be confined is shown hatched in figure 7.6(i). Plasticity is likely to be initiated from the middle of the plate where the bending moments are maximum. A total of eleven load increments from $\lambda=1.0$, 1.5, 1.75, 2.0, 2.1, 2.2, 2.3, 2.4, 2.5, 2.6 to 2.7 were used until collapse. Figure 7.6(ii) shows the spread of plasticity and figure 7.7 shows a plot of the load factor against the displacement of node 1. Plasticity is initiated when the load factor is 2.3.

7.3.2.1 Graph and table of CPU time

Figure 7.8 is a graph of the CPU time against the load factor for the TS, TSV, TSG and TSVG techniques for $\lambda=0.0$ to $\lambda=2.4$ only. This is to study the effect of the CPU times on the proposed techniques when only a few elements have become nonlinear as shown in figure 7.6(ii). Table 7.2 shows the total CPU time taken for each technique.

FINITE ELEMENT MESH

NO. OF ELEMENTS = 5
NO. OF NODES = 28

$E = 1.0 \times 10^7 \, lb/in^2$

$v = 0.24$

$\sigma_y = 16,000 \, lb/in^2$

$H' = 0.0$

$P = 100 \, lb/in^2$

Von Mises material

$B = 16$

Y-AXIS

X-AXIS

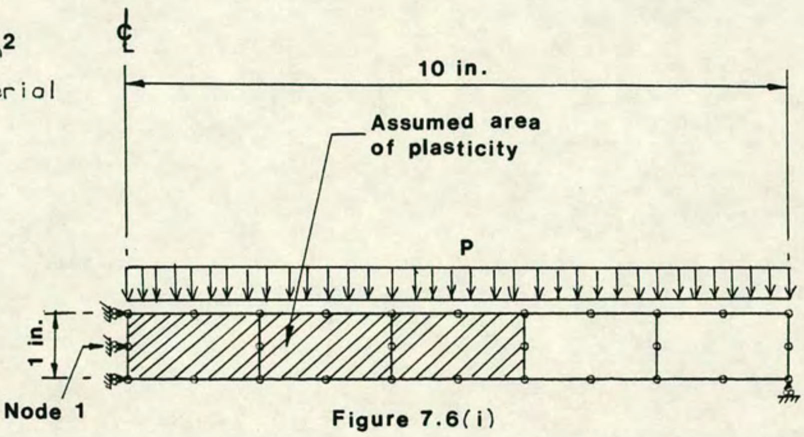


Figure 7.6(i)

FINITE ELEMENT MESH

NO. OF ELEMENTS = 5
NO. OF NODES = 28

$\lambda = 2.3$

$\lambda = 2.4$

Y-AXIS

X-AXIS

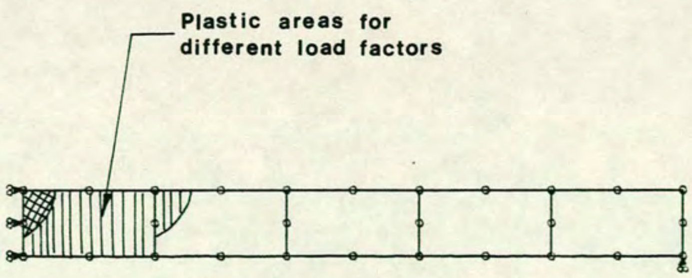


Figure 7.6(ii)

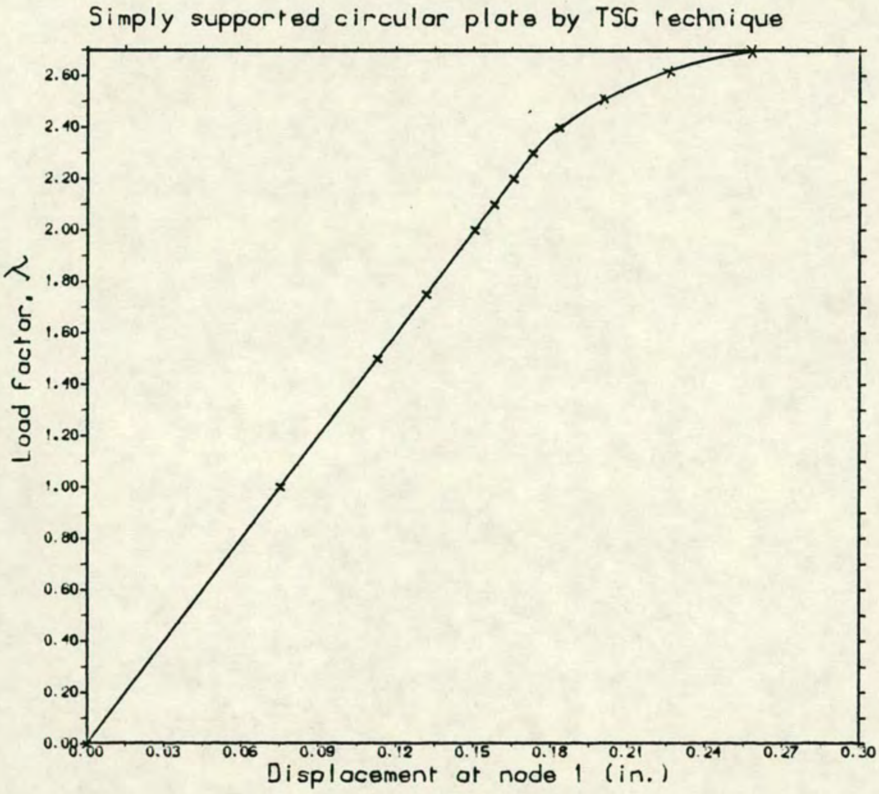


Figure 7.7

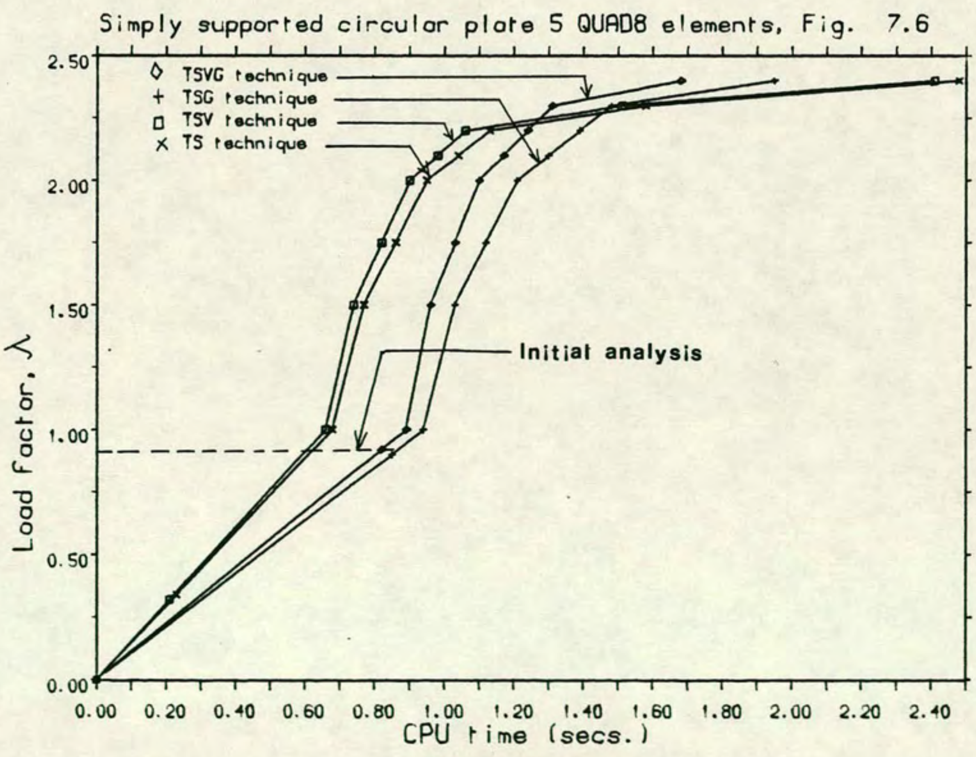


Figure 7.8

Simply supported circular plate 5 QUAD8 elements Figure 7.6			
Technique	Graph (Figure)	No. of iterations required	Total CPU time (secs.) for λ=0.0 to 2.4
IS	--	21	10.73
ISG	--	21	9.96
ISVG	--	21	9.90
TS	7.8	9	2.48
TSV	7.8	9	2.41
TSG	7.8	9	1.95
TSVG	7.8	9	1.68
ITS	--	19	3.06
ITSV	--	3	2.47
ITSG	--	19	3.66
ITSVG	--	3	1.82

Table 7.2 Total CPU time taken

7.3.2.2 Discussion of results

The table 7.2 and the graph shown in figure 7.8 again indicates that the TSG and TSVG techniques are the most efficient to $\lambda=2.4$. From figure 7.8, for $\lambda=0.0$ to 2.2, the theorems are inefficient because of the high cost of the initial analysis. However they begin to be efficient from $\lambda=2.2$ to 2.4. The size of the $[C_{aa}]$ matrix during this load increment is much smaller than the tangential stiffness matrix where only two elements have become plastic. This again suggests that efficiency is achieved when the nonlinearity is localised. It should also be noted that the variant forms of the theorems are also efficient. When the structure was loaded to collapse the area of plasticity is large and hence the theorems of geometric variation become inefficient.

7.3.3 Perforated plate

This is a classic example usually used for comparative studies in the use of finite elements[154,159] for plasticity analysis. The finite element mesh for a quarter of the plate using QUAD4 elements, material properties, dimensions and boundary conditions are as shown in figure 7.9(i). The problem was treated as a thin plate of unit thickness subject to plane stress. Two areas in which plasticity was assumed to be confined were considered in these comparative studies are shown hatched in figures 7.9(i) and (ii). The assumed areas are different, so that a study of the effects of the initial analysis on the total CPU time may be compared. A larger assumed area means a longer CPU time for the initial analysis.

The analysis was carried out using a total of eight load increments for $\lambda=0.56$, 0.66, 0.76, 0.86, 0.96, 0.98, 1.01 to 1.08. Figure 7.9(iii) shows the spread of plasticity and figure 7.10 is a graph of the load factor against the displacement of node 1. Plasticity is initiated at the first load increment.

FINITE ELEMENT MESH

NO. OF ELEMENTS = 78
NO. OF NODES = 98

$E = 7000 \text{ kg/mm}^2$

$v = 0.20$

$\sigma_y = 24.3 \text{ kg/mm}^2$

$H' = 225.0 \text{ kg/mm}^2$

$P = 12.15 \text{ kg/mm}^2$

Von Mises material

B = 18

Y-AXIS

X-AXIS

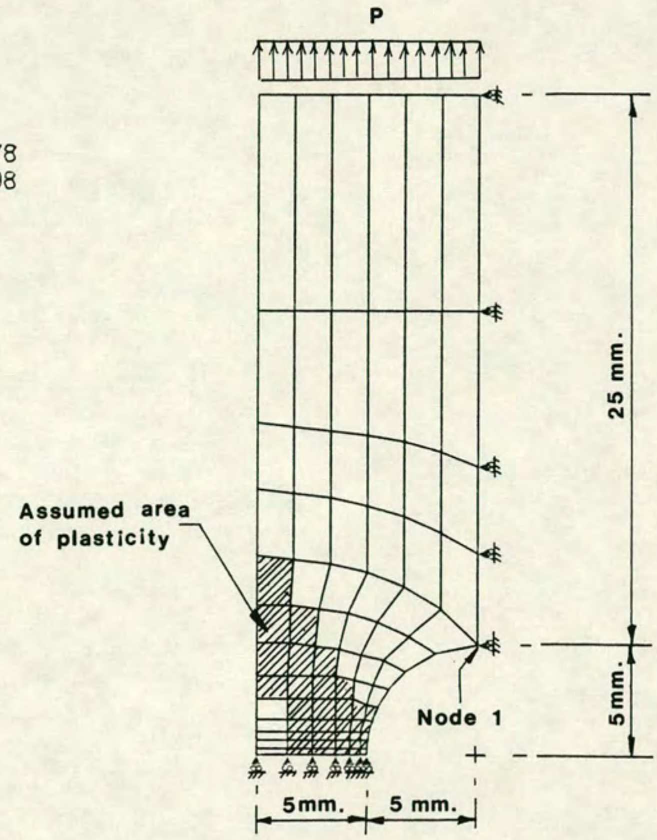


Figure 7.9(i)

FINITE ELEMENT MESH

NO. OF ELEMENTS = 78
NO. OF NODES = 98

Y-AXIS

X-AXIS

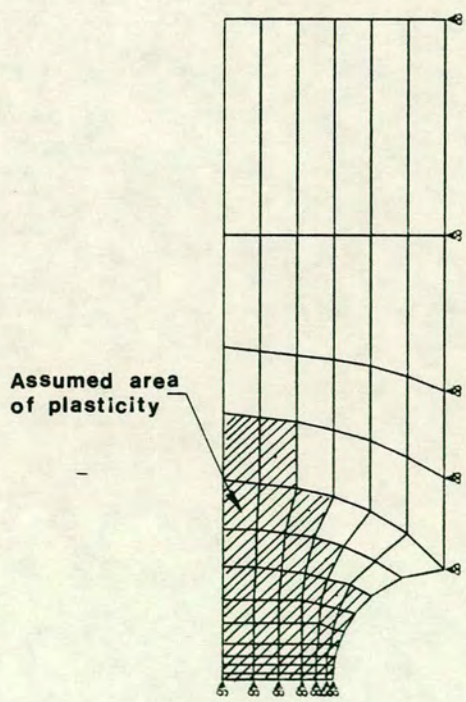
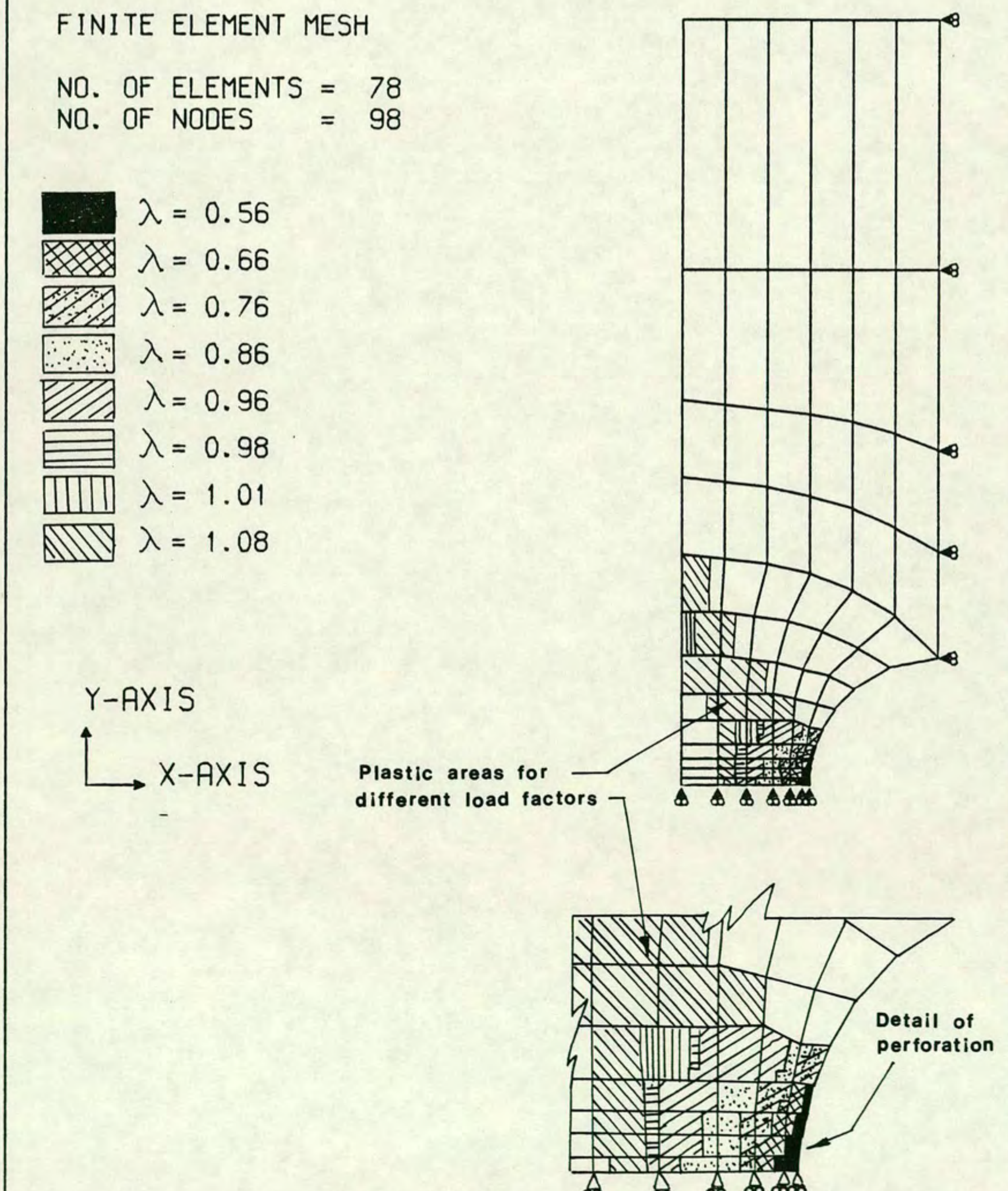
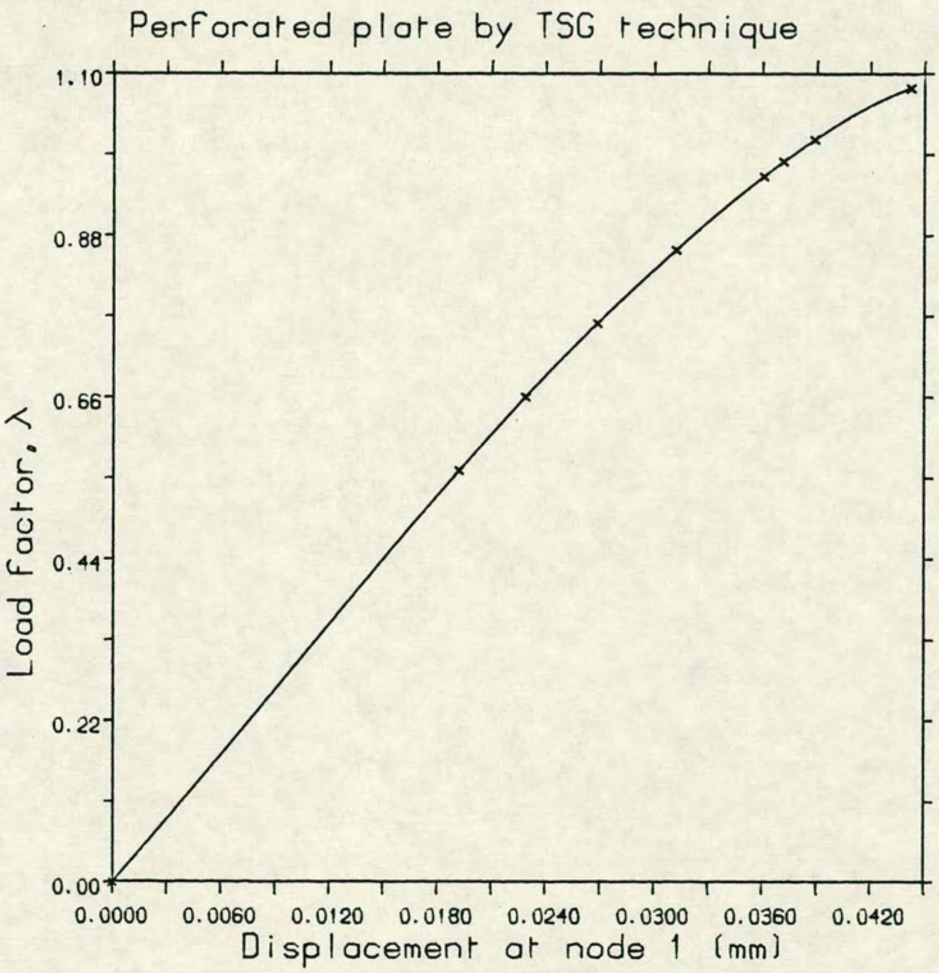


Figure 7.9(ii)



. Figure 7.9(iii)



Perforated plate by TSG technique
1.10
0.88
0.66
0.44
0.22
0.00
0.0000
0.0060
0.0120
0.0180
0.0240
0.0300
0.0360
0.0420
Load factor, λ
Displacement at node 1 (mm)

Figure 7.10

7.3.3.1 Table of CPU time

The total CPU time taken for each assumed area of plasticity and for each technique are tabulated in table 7.3.

Perforated plate 78 QUAD4 elements Figure 7.9			
Technique	Figure	No. of iterations required	Total CPU time (secs.) for λ=0.0 to 1.08
IS	--	69	113.12
ISG	7.9(i)	69	116.37
	7.9(ii)	69	119.24
ISVG	7.9(i)	69	116.04
	7.9(ii)	69	118.90
TS	--	17	83.33
TSV	--	23	111.35
TSG	7.9(i)	17	58.67
	7.9(ii)	17	59.76
TSVG	7.9(i)	23	67.22
	7.9(ii)	23	68.35
ITS	--	35	82.60
ITSV	--	30	100.49
ITSG	7.9(i)	35	78.47
	7.9(ii)	35	83.20
ITSVG	7.9(i)	30	71.66
	7.9(ii)	30	76.36

Table 7.3 Total CPU time taken

7.3.3.2 Discussion of results

The ITSG technique is only efficient when the areas of plasticity may be assumed confined to relatively few elements as in figure 7.9(i). If the number of elements are greater as for example shown in figure 7.9(ii), then the efficiency of this technique deteriorates. However the ITSVG, TSG and TSVG techniques are still very efficient and are not sensitive to the effects of the initial analysis. This is because with these techniques the initial analysis is only a small proportion of the total CPU time taken.

7.3.4 Notched beam

The notched beam was analysed in reference [112,158]. The finite element mesh for half the beam using QUAD8 elements, material properties, dimensions and boundary conditions are as shown in figure 7.11(i). The beam is of unit thickness and plane stress conditions were assumed. Again two areas, in which the plasticity was assumed to be confined were considered in these comparative studies. These areas are shown in figures 7.11(i) and (ii). This example is to study the effect of a structure with large semi-bandwidth (in this case it is 68 compared to 236 degrees of freedom) in using the theorems of geometric variation as a nonlinear analysis technique.

The analysis was carried out in five load increments for λ=1.0, 3.0, 4.5, 5.5 to 6.0. Figure 7.11(iii) shows the elements which have become plastic during the load increments, and figure 7.12 is a plot of the load factor against the displacement of node 1.

7.3.4.1 Table of CPU time

The total CPU time taken for each assumed area of plasticity and for each analysis are tabulated in table 7.4.

FINITE ELEMENT MESH

NO. OF ELEMENTS = 31
NO. OF NODES = 118

$E = 3 \times 10^7 \, lb/in^2$

$v = 0.28$

$\sigma_y = 6 \times 10^4 \, lb/in^2$

$H' = 0.0$

$F = 536$ lb.

Von Mises material

$B = 68$

Y-AXIS
X-AXIS

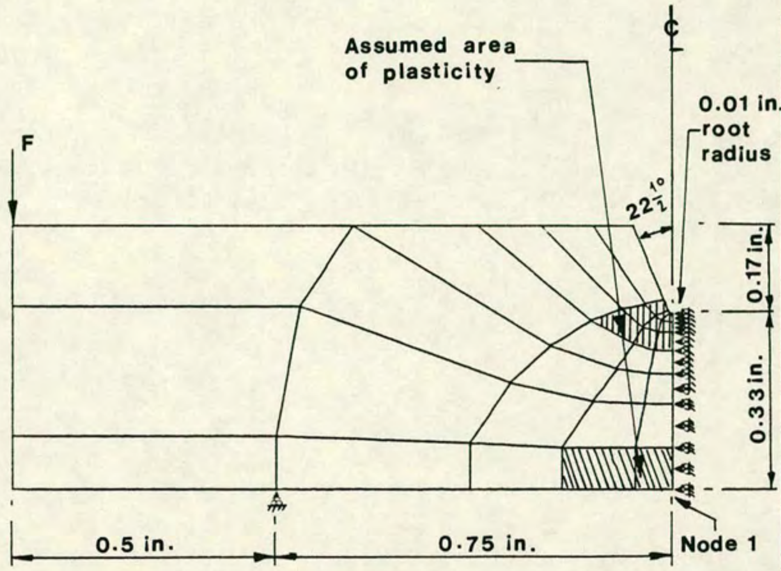


Figure 7.11(i)

FINITE ELEMENT MESH

NO. OF ELEMENTS = 31
NO. OF NODES = 118

Y-AXIS
X-AXIS

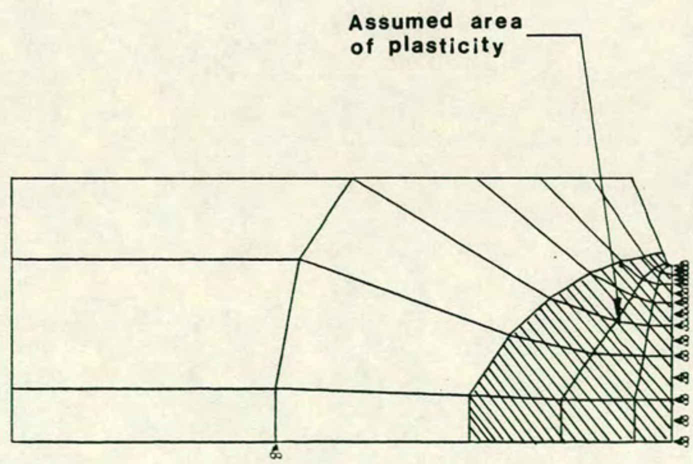


Figure 7.11(ii)

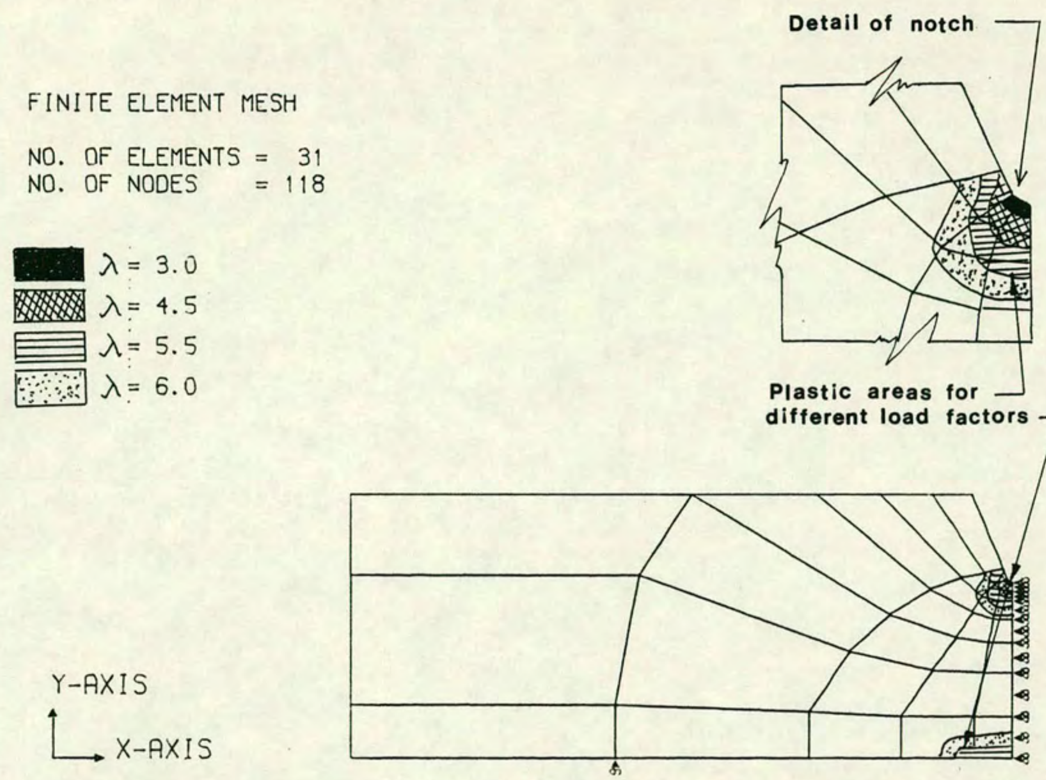


Figure 7.11(iii)

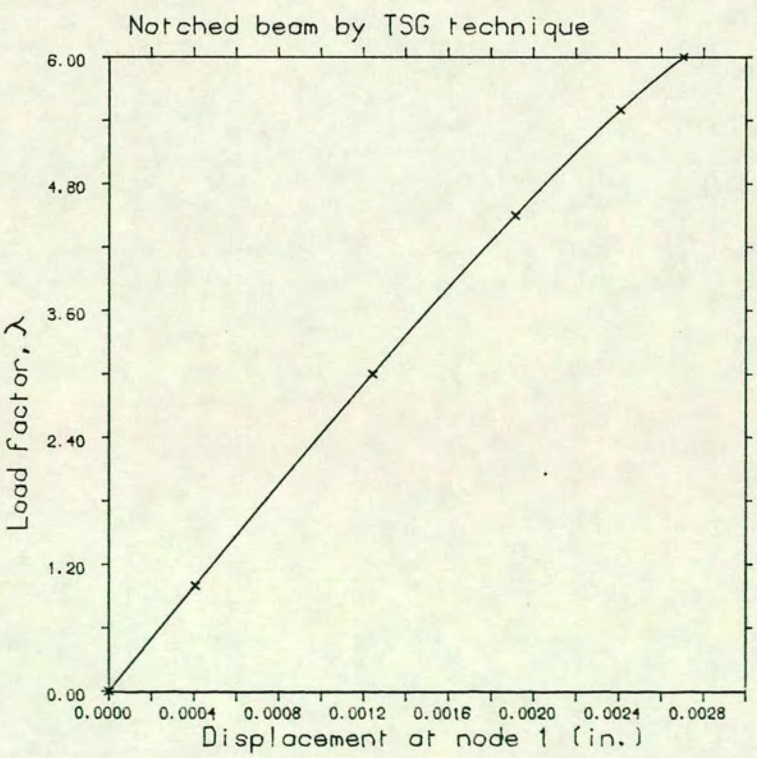


Figure 7.12

Notched beam 31 QUAD8 elements Figure 7.11			
Technique	Figure -	No. of iterations required	Total CPU time (secs.) for λ=0.0 to 6.0
IS	--	23	55.98
ISG	7.11(i)	23	60.90
	7.11(ii)	23	67.63
ISVG	7.11(i)	23	60.83
	7.11(ii)	23	67.49
TS	--	10	142.94
TSV	--	12	170.76
TSG	7.11(i)	10	56.10
	7.11(ii)	10	61.41
TSVG	7.11(i)	12	57.57
	7.11(ii)	12	62.89
ITS	--	12	85.55
ITSV	--	13	136.47
ITSG	7.11(i)	12	51.44
	7.11(ii)	12	56.40
ITSVG	7.11(i)	13	110.00
	7.11(ii)	13	115.43

Table 7.4 Total CPU time taken

7.3.4.2 Discussion of results

As may be seen in table 7.4, for a structure with large semi-bandwidth the initial analysis for the assumed area is only a small proportion of the total CPU time taken. The theorems of geometric variation by the TSG and TSVG techniques are particularly efficient because the reduction of the $[C_{aa}]$ matrix takes less time than for the tangential stiffness matrix at each iteration. This is because the plasticity only affects a few elements. Substantial savings in CPU time (of approximately a half) may be achieved by using the TSG and TSVG techniques, when the semi-bandwidth of a structure is large.

7.4 Summary

A reanalysis technique based on the theorems of geometric variation has been investigated for the efficient analysis of material nonlinear problems. Several examples were given to indicate the particular areas of where the theorems will be efficient. These are when the areas of plasticity are small and localised, and the semi-bandwidth of the structure is large. If a problem is analysed until collapse, it is very likely that large areas will be plastic. In this case the theorems will be inefficient at the later stages of the load incrementation.

The various techniques of Newton-Raphson using the theorems of geometric variation, show that the tangential stiffness technique is the most efficient. This is because for this particular technique the matrix to be reduced only involves the portion of the structure that is changing. This is in contrast to the usual procedure where the tangential stiffness matrix is reduced for the whole structure. The use of the initial stiffness technique with the theorems does not have any advantage in terms of CPU time. While the savings in the CPU time for the initial/tangential stiffness technique are not considerable. Therefore the most promising technique that was proposed in Chapter 4, is the tangential stiffness technique by the theorems of geometric variation. This was clearly demonstrated in the final example of section 7.3. Finally it should be noted that the proposed techniques are very much dependent on the experience and intuition of the analyst in deciding the areas to which the plasticity will be confined.

CHAPTER 8

EFFICIENCY OF THE THEOREMS OF GEOMETRIC VARIATION FOR GEOMETRICAL NONLINEAR PROBLEMS

8.1 Introduction

This chapter will investigate the efficiency of the theorems of geometric variation for the analysis of geometrical nonlinear problems.

The proposed techniques of Chapter 4 were tested for efficiency in comparison with each other and that of the Newton-Raphson method and its degenerate forms. These comparisons are based on the CPU time taken for the analysis of some geometrical nonlinear problems. These problems will also indicate the limitations of the proposed techniques as a nonlinear solution algorithm.

8.2 Geometrical nonlinear analysis by the theorems of geometric variation

When deformations become large compared to the dimensions of the structure, the problem is no longer linear. The coefficients of the stiffness matrix are now dependent on the current state of displacements and the resulting equations must be solved iteratively. In this chapter, the Total Lagrangian coordinate system is adopted which coincides with the initial undeformed position of the body. The material is assumed elastic (and hence the strains small) but the displacements may be large. The stress and strain measures used are the 2nd. Piola-Kirchoff stress and Green's strain respectively. The derivation of the tangential stiffness matrix for such problems have been outlined in Chapter 2. As in Chapter 7, the most important part of the proposed techniques is in the evaluation of the compensation forces. This is given by equation (4.13) of Chapter 4. The analysis is now briefly described as follows.

8.2.1 Initial analysis

The geometrical nonlinear analysis of a structure by the theorems may be undertaken by first assuming which parts of the structure is likely to behave nonlinearly. These are shown hatched in the figures of the examples studied in this chapter. Unit load analyses are performed at each degree of freedom of the nodes of the assumed nonlinear elements. In addition unit load analysis at the degree of freedom of the nodes where there are applied loadings must also be undertaken. These are the 'a' affected nodes and 'u' unaffected nodes respectively. This analysis is given by:

$$[K_o][\delta_a \mid \delta_u] = [I_a \mid I_u] \tag{8.1}$$

The assumption concerning which part of the structure is likely to be nonlinear requires experience and careful physical evaluation. This evaluation is more difficult in geometrical nonlinear problems compared to material nonlinear problems. As mentioned in reference [122], examples in which the geometrical nonlinear behaviour is localised are very scarce in the literature.

To use the theorems of geometric variation effectively, some plausible criteria must be employed to detect which elements are behaving nonlinearly. In material nonlinearity of Chapter 7, the criterion was the presence of plastic strains to indicate that the element is nonlinear. In geometrical nonlinearity no such criterion exists, therefore it was felt that the displacements could be used as a criterion. For example, if the nonlinear analysis of a plate is required then the displacements in the direction of the plate thickness should not exceed a specified fraction or factor of the plate thickness. Henceforth this factor will be termed the displacement factor, d_f. The value of d_f chosen in the analysis will be dependent on the analyst. If d_f is set at some very small value then the whole structure will be analysed nonlinearly which is the normal procedure of geometrical nonlinear analysis. If it is some very large value then the criterion will never be satisfied and the analysis is linear. Somewhere in between these extremes, some elements will be modelled as linear or nonlinear. The theorems may then be used efficiently in such situations as shown in the example problems presented later.

8.2.2 Analysis of modified structure

The response of the modified structure may now be obtained after the initial unit load analyses have been performed. If an element has become nonlinear because the displacements at its nodes exceed the specified value of d_f, then its new stiffness (from equation (2.33)) may be expressed as:

$$[K^e_T] = [K^e_0] + [K^e_L] + [K^e_\sigma] \qquad (8.2)$$

In equation (8.2) $[K^e_T]$ is the tangential stiffness of the element which accounts for large displacements. It must be regularly evaluated during the analysis because the element undergoes large deformations. The change in element stiffness is therefore given by:

$$\begin{aligned}[\Delta K^e] &= [K^e_T] - [K^e_0] \\ &= ([K^e_0]+[K^e_L]+[K^e_\sigma]) - [K^e_0] \\ &= [K^e_L] + [K^e_\sigma] \qquad (8.3)\end{aligned}$$

Equation (8.3) may now be used with equation (4.13) to form the matrix of compensation forces for each nonlinear element. This is simply given by:

$$[C^e_{aa} \mid C^e_{au}] = ([K^e_L]+[K^e_\sigma])\ [\delta^e_a \mid \delta^e_u] \qquad (8.4)$$

The overall matrix of compensation forces defined by equations (4.12) or (4.15) may be formed by adding each contribution of $[C^e_{aa} \mid C^e_{au}]$ from a nonlinear element. As explained in Chapter 4 the formation of the overall matrix of compensation forces will depend on the technique selected for the analysis.

As shown by Wood[150] the initial stiffness technique is not particularly successful in geometrical nonlinear problems. The results of this technique will not be given in section 8.3. However the technique has been tried on all the examples of section 8.3, and was found to be frequently unreliable for stiffening problems. The solution obtained tend to diverge after a few load increments. This indicate that the use of the initial elastic stiffness matrix is not sufficient to solve geometrical nonlinear problems. The matrix must be updated to account for the large displacements for such problems.

8.3 Investigation of the efficiency

Four benchmark examples were used to investigate the efficiency of the proposed techniques of Chapter 4 for geometrical nonlinear problems. These are the analyses for: a clamped circular plate; a cantilever beam; a shallow spherical shell; and a shallow circular arch.

The measure of efficiency is based on the CPU time taken for each analysis for different values of the displacement factor, d_f. The first test is a comparison between the various proposed techniques such that the most efficient one may be identified. The second test is a comparison with the Newton-Raphson method and its degenerate forms. This comparison will indicate whether the proposed techniques are suitable for use in geometrical nonlinear problems.

8.3.1 Clamped circular plate

This problem was analysed by Weil and Newmark[147] using the Ritz procedure and subsequently by Wood[150] using the finite element method. In this finite element analysis it is treated as an axisymmetric problem. The finite element mesh for half the plate, dimensions, material properties and boundary conditions are as shown in figure 8.1. A total of 15 load increments with iterations within each load increment were used and the problem was solved using the Newton-Raphson methods and the theorems of geometric variation.

To use the theorems an initial analysis for a number of unit loads is required. The assumed nonlinear elements are shown hatched in figure 8.1. This assumed nonlinear area is reasonable because of the likelihood that the whole structure may behave nonlinearly. The chosen displacement factors were, d_f=0.0, 0.2, 0.4, 0.6, 0.8, 1.0 and 2.6. The results obtained are plotted as load factor against the displacement of node 1 in figure 8.2. For d_f=0.0, the results were almost identical to the Ritz solution of Weil and Newmark[147]. The plate exhibits stiffening behaviour as the load is incremented. For small values of d_f, the results approximates those for d_f=0.0. With a higher value of d_f the analysis tends towards a linear one because very few elements

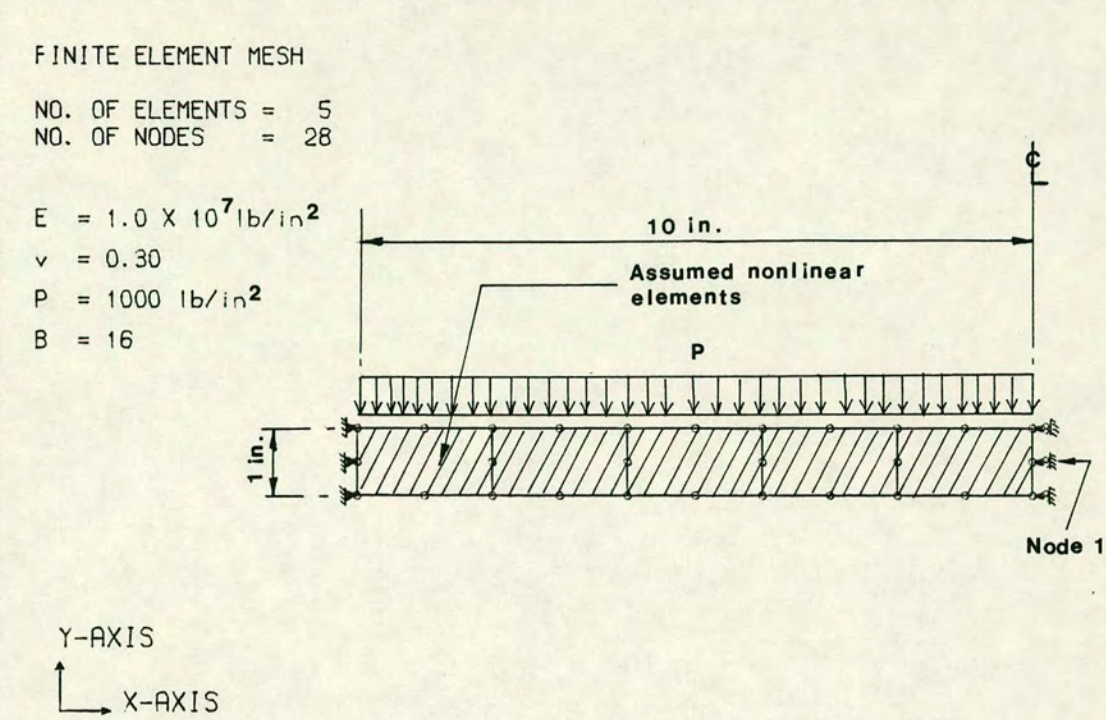


Figure 8.1

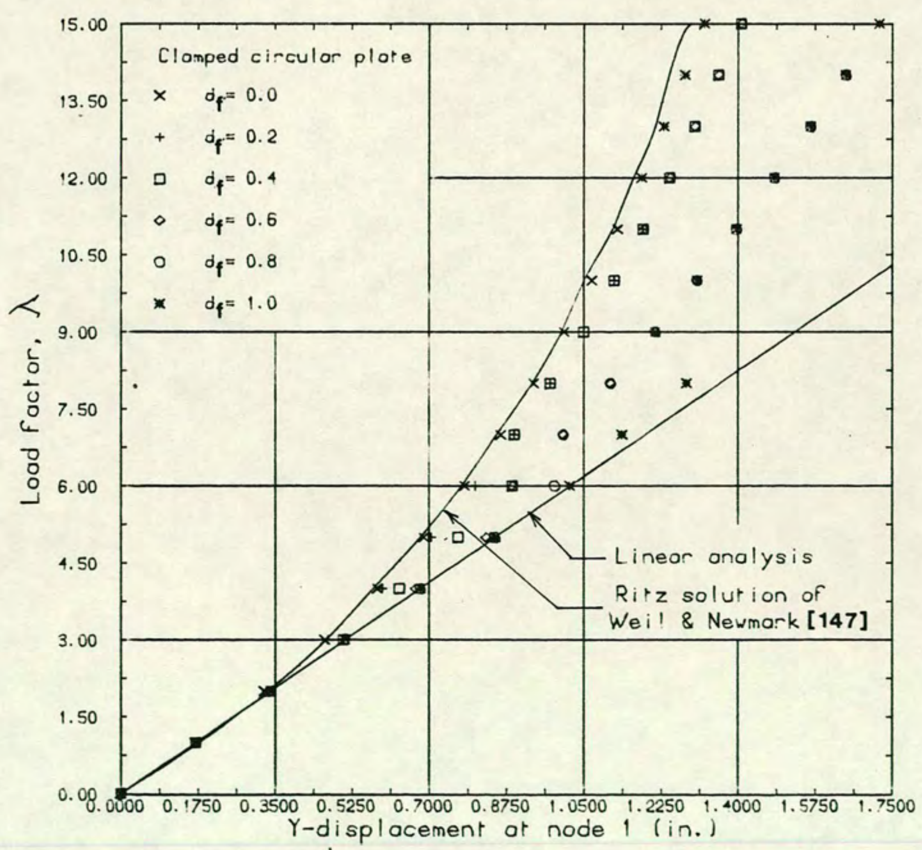


Figure 8.2

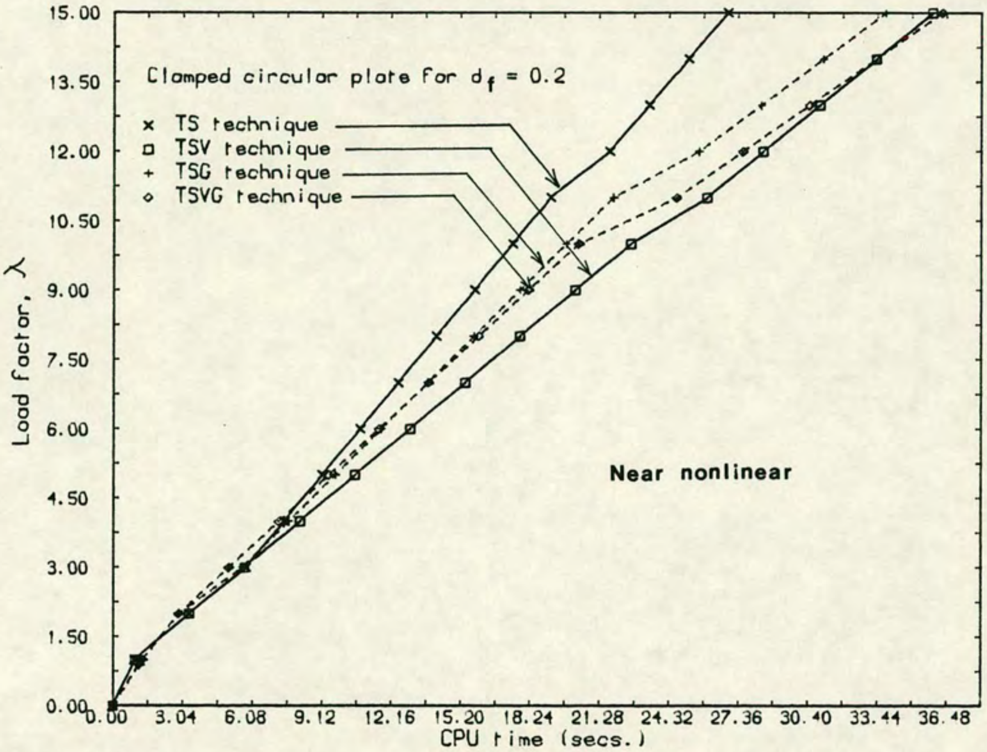


Figure 8.3(i)

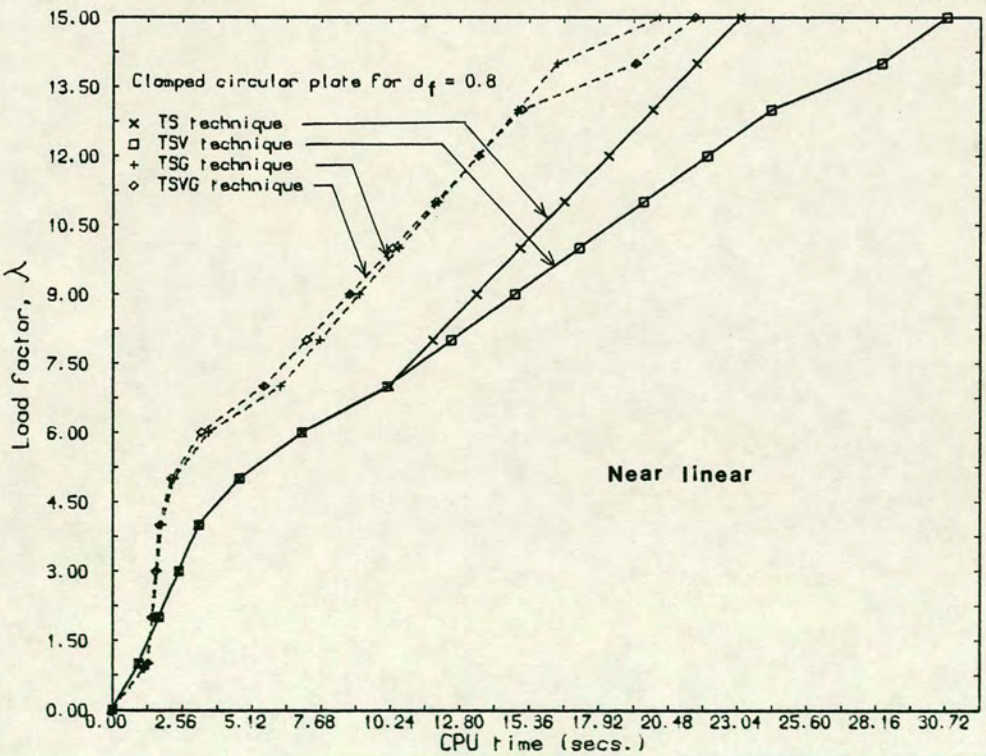


Figure 8.3 (ii)

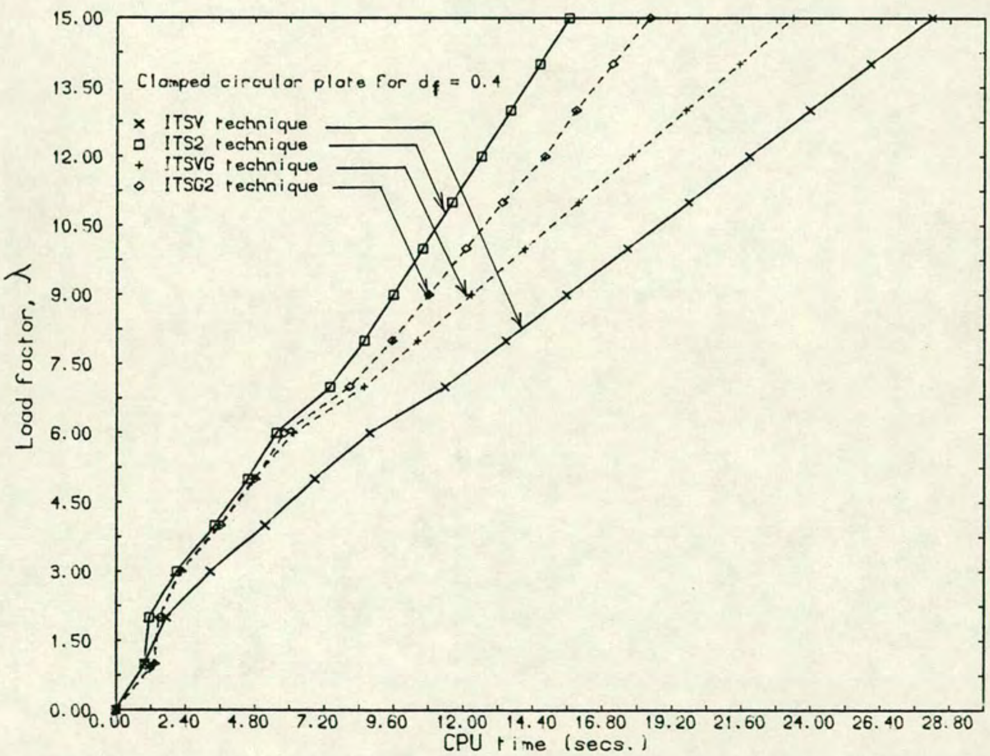


Figure 8.3 (iii)

have satisfied the nonlinear criterion.

8.3.1.1 Graphs and table of CPU time

Figures 8.3(i) and (ii) are graphs of the load factor against the CPU time taken for the TS, TSV, TSG and TSVG techniques only. These graphs give the CPU time during the loading history of the structure. The graph of figure 8.3(iii) includes two additional new techniques called ITS2 and ITSG2. A description of these new techniques is given below:

- **ITS2 technique** : This technique is similar to that of the ITS technique which uses a combination of the initial and tangential stiffness techniques. However the stiffness is only updated at the second iteration of each load increment. It is then kept constant throughout the load increment until the second iteration of the next load increment. This technique is described in section 2.4.1.3 and used for comparison with the ITSG2 technique.

- **ITSG2 technique** : The version of the ITS2 technique using the theorems of geometric variation to solve the nonlinear equations. It is described in section 4.2.2.3.

Table 8.1 shows the total CPU time taken for each different technique and value of d_f used in the analysis.

Clamped circular plate 5 QUAD8 elements in figure 8.1 For λ= 0.0 to 15.0							
	CPU time for each value of d_f (seconds)						
Technique	d_f =0.0	d_f =0.2	d_f =0.4	d_f =0.6	d_f =0.8	d_f =1.0	d_f =2.6
TS	25.99	26.93 †	25.56	26.00	24.86 ‡	22.22	11.25
TSG	40.82	33.83 †	26.98	24.10	20.12 ‡	16.05	3.51
TSV	36.45	35.87 †	32.88	31.84	30.79 ‡	26.55	11.25
TSVG	43.46	36.28 †	27.52	24.50	21.48 ‡	15.89	3.28
ITS	19.87	Diverges	Diverges	Diverges	Diverges	Diverges	11.29
ITSG	28.41	Diverges	Diverges	Diverges	Diverges	Diverges	3.74
ITSV	30.94	29.68	28.14 *	27.34	26.00	23.21	11.30
ITSVG	32.19	27.58	23.36 *	21.64	19.08	15.29	3.73
ITS2	16.18	Diverges	15.39 *	15.43	15.13	13.04	3.22
ITSG2	23.66	Diverges	17.27 *	16.28	14.94	13.33	3.74
No. of nonlinear elements at λ=15	5	5	4	4	4	3	0

Table 8.1 Total CPU time taken

† See graph in figure 8.3(i)

‡ See graph in figure 8.3(ii)

* See graph in figure 8.3(iii)

8.3.1.2 Discussion of results

It is obvious from table 8.1 that for small values of d_f the theorems of geometric variation are inefficient for all the proposed techniques. As d_f increases it becomes more efficient, although for the ITSG2 technique there were no significant improvements. The second observation is that as d_f increases the CPU time taken decreases. This is because the number of iterations (not given in the table) required during each load increment decreases. During the first few load increments, the criterion d_f is not satisfied and hence the analysis is linear resulting in a decrease in CPU time.

From the graph of figure 8.3(i), the TSG and TSVG techniques (for d_f=0.2) are shown to be efficient at low load factors because few elements behave nonlinearly. At higher load factors only the TSVG technique is efficient. For figure 8.3(ii) the TSG and TSVG techniques (for d_f=0.8) are efficient at all stages of loading because very few elements are nonlinear. The ITSVG technique is inefficient in comparison to the ITSG2 technique as may be seen in figure 8.3(iii) and table 8.1 for d_f=0.4.

It is difficult to suggest which is the best choice of solution algorithm and the factor d_f from this example. These are probably very much dependent on the type of problem and experience of the analyst. For d_f=0.4 to 2.6, the ITSVG and ITSG2 techniques are the most efficient (see table 8.1) but as will be shown later, these techniques are not stable for all problems. However if the analysis is undertaken using the TSG or TSVG technique with a high value of d_f, then the technique would be competitive to the usual procedure. The efficiency of these two techniques may be inferred from table 8.1 for values of d_f=0.4 to 2.6. The TSG and TSVG techniques are the most stable techniques as will be discussed later.

8.3.2 Cantilever beam

The geometrical nonlinear finite element analysis of this beam was first undertaken by Bathe et.al.[16]. The beam was divided into five QUAD8 elements as shown in figure 8.4. The material properties, dimensions and boundary conditions are also shown in the figure. Plane stress conditions were assumed in the analysis. The beam was of unit thickness and was loaded by incrementing the load factor, λ. A total of 20 load increments were applied with iterations within a load increment.

To use the theorems of geometric variation efficiently, an initial analysis for a number of unit loads is required. The assumed nonlinear elements are shown hatched in figure 8.4. The factors d_f used for the analysis were 0.0, 0.5, 1.0, 1.5, 2.0 and 2.5. For d_f=0.0 all the elements will behave nonlinearly. For d_f=0.5 to 2.5, only some elements will behave nonlinearly depending on whether their nodal displacements exceed that of d_f. The choice of d_f depends on the analyst and type of analysis required.

The method of solution used are the TS, TSV, TSG and TSVG techniques. The displacements are plotted against load factors in figure 8.5. As expected, for d_f=0.0 the results obtained were in excellent agreement with those of Bathe's et.al.[16]. The beam exhibits stiffening behaviour as the load increases because of the membrane forces developing throughout the beam. For d_f=0.5 to 2.5 the response only approximates the true nonlinear behaviour because the nonlinear effects of displacements are ignored in the elements near the fixed support. The elements here are assumed to behave linearly because the nodal displacements never exceed the value of d_f chosen in the analysis. The smaller the value of d_f, the closer are the results to the true nonlinear solution. Only when d_f=0.0 do the techniques predict the geometrically nonlinear behaviour accurately.

8.3.2.1 Graphs and table of CPU time

Figures 8.6(i) and (ii) are graphs of the load factor against the CPU times taken for a particular value of d_f. The graphs should be considered in conjunction with table 8.2, where the total CPU time for each value of d_f are tabulated.

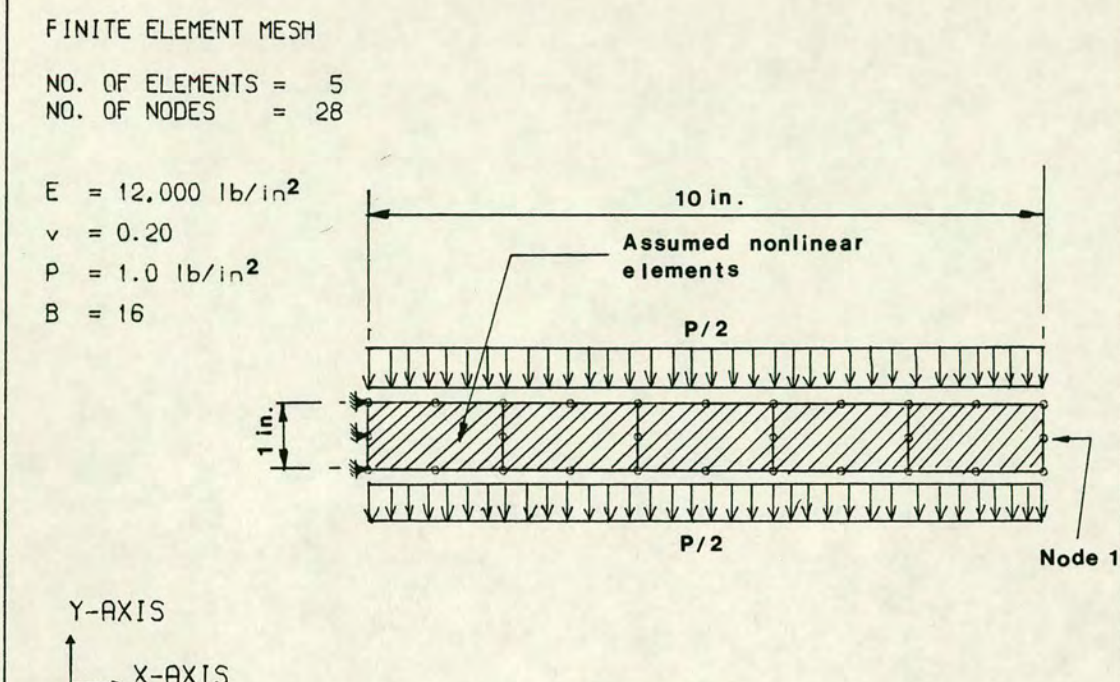


Figure 8.4

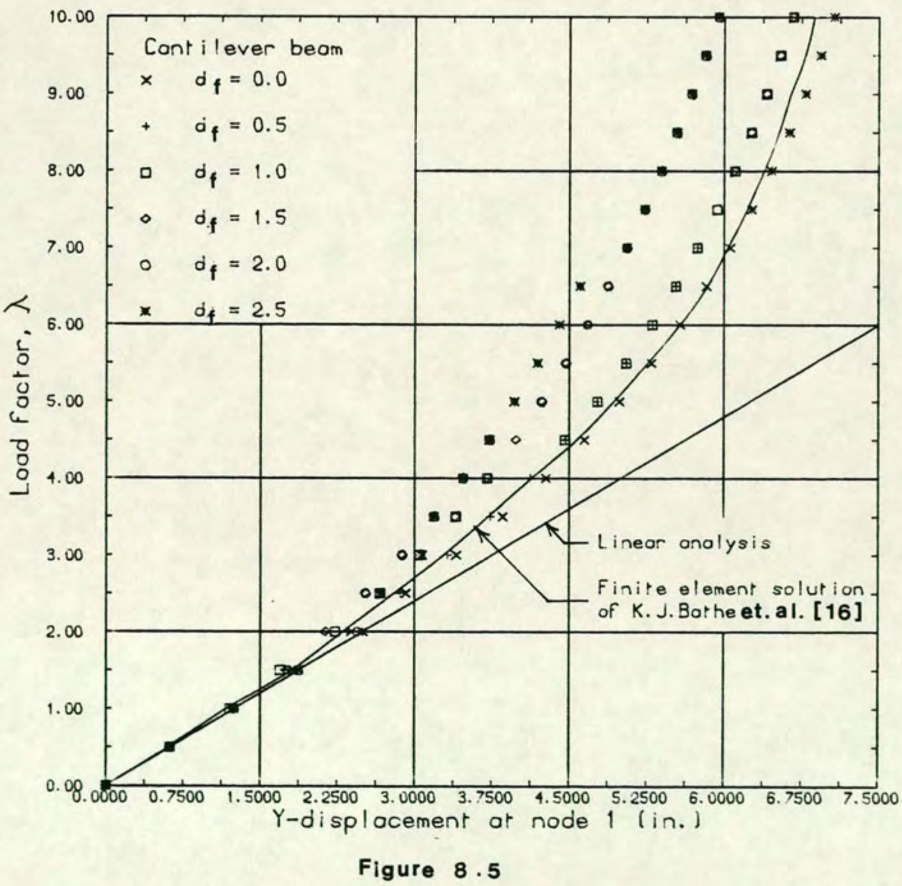


Figure 8.5

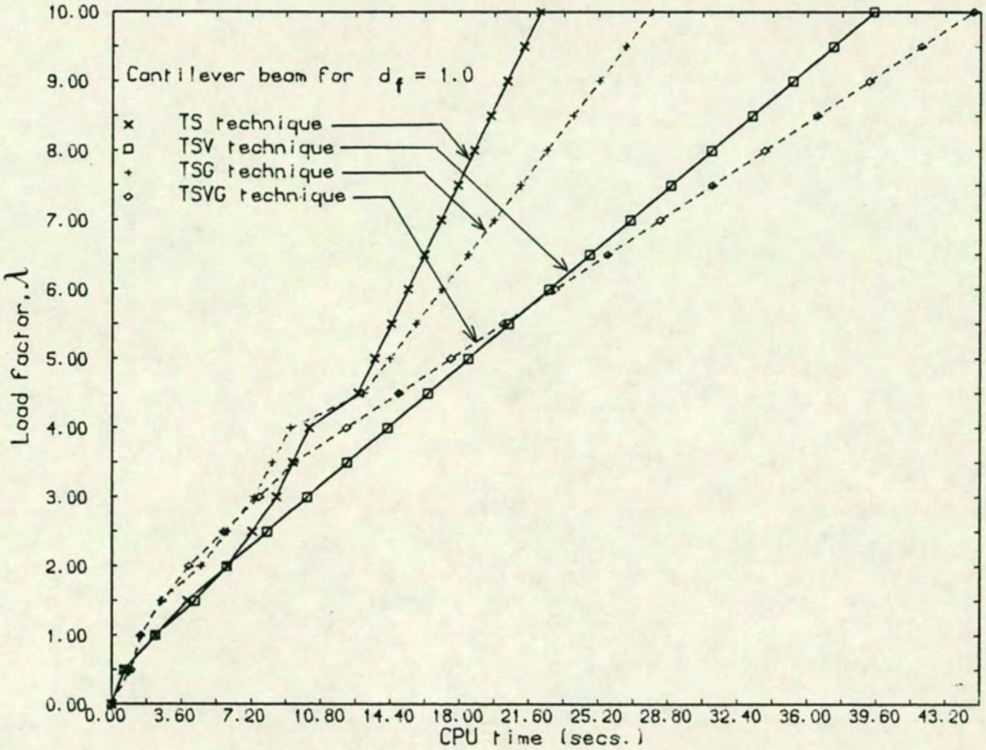


Figure 8.6(i)

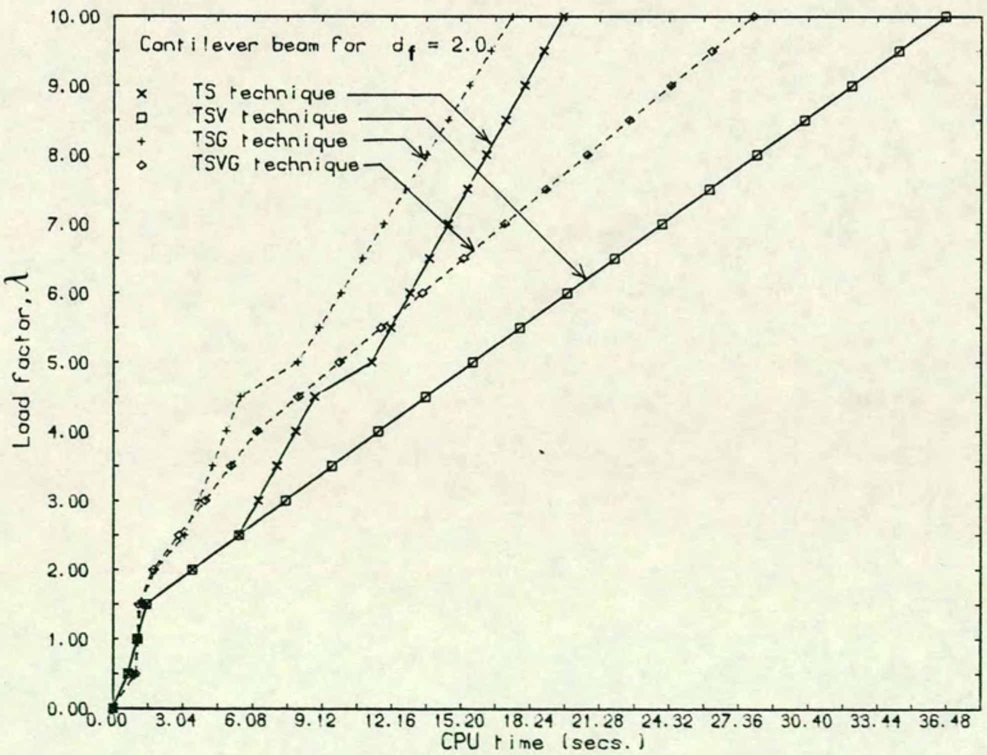


Figure 8.6 (ii)

Cantilever beam 5 QUAD8 elements in figure 8.4 For λ= 0.0 to 10.0						
	CPU time for each value of d_f (seconds)					
Technique	d_f =0.0	d_f =0.5	d_f =1.0	d_f =1.5	d_f =2.0	d_f =2.5
TS	20.75	22.05	22.15 †	22.36	19.68 ‡	19.99
TSG	44.61	35.78	28.00 †	23.99	17.46 ‡	15.94
TSV	38.27	40.36	39.51 †	39.23	36.42 ‡	35.04
TSVG	68.09	57.65	44.69 †	37.83	28.01 ‡	24.58

Table 8.2 Total CPU time taken

† See graph in figure 8.6(i)

‡ See graph in figure 8.6(ii)

8.3.2.2 Discussion of results

The stability of the ITS and ITS2 techniques cannot be guaranteed for this problem (see section 8.4).

Table 8.2 does not quite reveal the whole story. The graphs in figures 8.6 (i) and (ii) give the total CPU time during the load incrementation. For d_f=1.0 and at low load factors, the theorems are efficient because the number of nonlinear elements is small. As the load increases more elements become nonlinear and the efficiency begins to drop. At very low load factors, λ=0.0 to 0.5 the theorems are also inefficient because the initial analysis time is a large proportion of the total CPU time taken, although the size of $[C_{aa}]$ is small. For d_f=2.0, the theorems are efficient at every stage except at low load factors. The variant forms take more time because they require more iterations.

8.3.2.3 Effect of the semi-bandwidth

To study the effect of the semi-bandwidth on the solution techniques, a finer mesh of elements was used as shown in figure 8.7. The assumed area of nonlinearity is shown hatched in the figure. The nodes were numbered along the shorter side which gives a semi-bandwidth, B of 22. When numbered along the longer side the semi-bandwidth was 70. These two numbering systems were used to study the effects of bandwidth on the efficiency of the theorems. The results of the CPU time taken are plotted in figures 8.8(i) and (ii), and tabulated in table 8.3.

Cantilever beam 20 QUAD8 elements in figure 8.7 For λ= 0.0 to 10.0						
	CPU time for each value of d_f (seconds)					
	d_f =1.0		d_f =1.5		d_f =2.0	
Technique	B=22	B=70	B=22	B=70	B=22	B=70
TS	126.18 †	482.77 ‡	122.96 †	473.71 ‡	118.29	457.34
TSG	309.57 †	321.63 ‡	216.45 †	228.28 ‡	169.67	182.11
TSV	199.04.	765.82.	195.85.	756.76.	180.52	700.54
TSVG	438.34.	450.14.	334.04.	345.89.	232.53	244.28

Table 8.3 Total CPU time taken

† See graph in figure 8.8(i)

‡ See graph in figure 8.8(ii)

From table 8.3, the TS and TSV techniques are greatly affected by the semi-bandwidth of $[K_T]$. As expected the larger the semi-bandwidth the longer is the CPU time taken for the TS and TSV techniques. The theorems on the otherhand are only dependent on the initial analysis

FINITE ELEMENT MESH

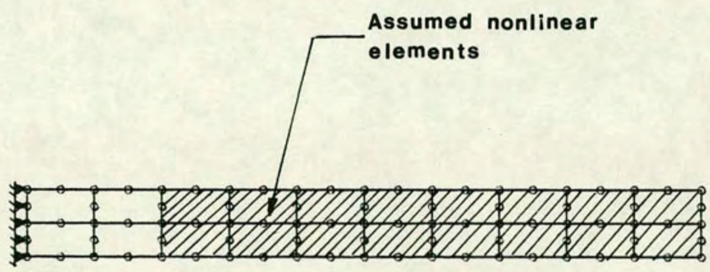
NO. OF ELEMENTS = 20
NO. OF NODES = 85

$E = 12,000\ lb/in^2$

$\nu = 0.20$

$P = 1.0\ lb/in^2$

$B = 22$ or 70



Y-AXIS

X-AXIS

Figure 8.7

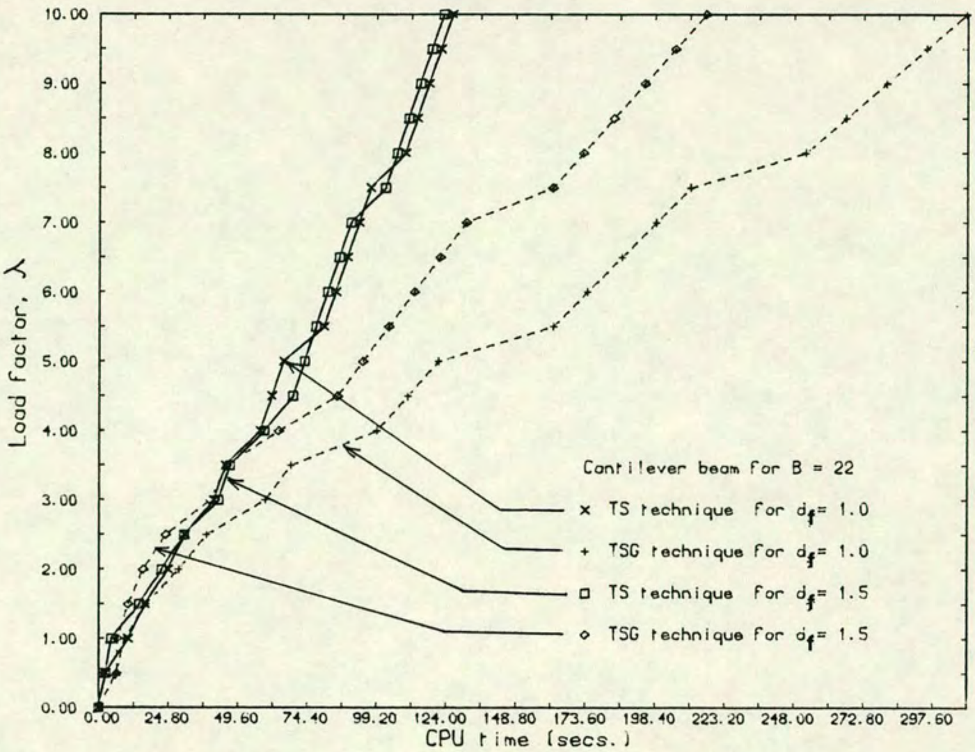


Figure 8.8(i)

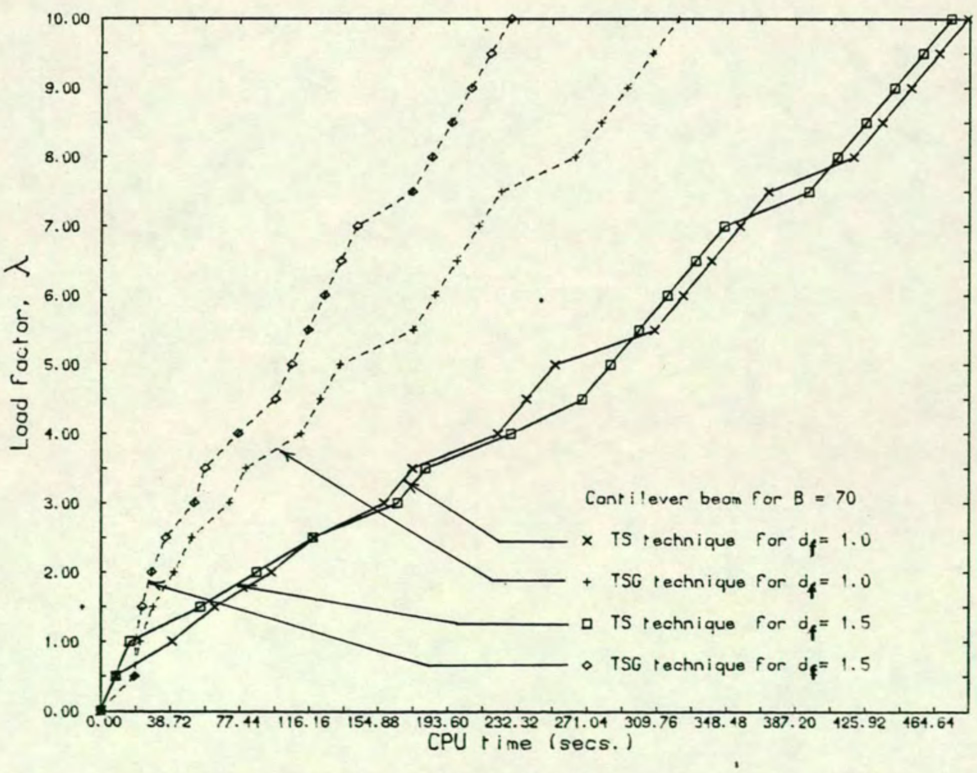


Figure 8.8(ii)

and the reduction of $[C_{aa}]$. The $[C_{aa}]$ matrix is independent of the semi-bandwidth and its reduction takes the same time whether B is 22 or 70. The size of the $[C_{aa}]$ matrix only depends on the number of nonlinear elements.

In the graphs of figure 8.8(i) and (ii), the theorems are shown to be inefficient at low load factors as before. For B=70, the theorems are very efficient in comparison to B=22 regardless of the load factor (except for very low load factors). Thus the theorems of geometric variation are efficient when the semi-bandwidth of a structure is large and the nonlinearity is localised. This confirms the earlier observations of the technique discussed in Chapters 6 and 7.

8.3.2.4 Summary of results

From the graphs and tables it is obvious that for d_f=0.0, the theorems are inefficient in comparison to the usual procedures. This is because, at this load factor all elements behave nonlinearly and the size of matrix $[C_{aa}]$ is the same order as $[K_T]$. However $[C_{aa}]$ is full and unsymmetric and therefore it is more expensive to reduce than $[K_T]$. As the value of d_f increases, the number of nonlinear elements gradually decreases and the theorems of geometric variation become efficient.

8.3.3 Shallow spherical shell

This is a common example used in geometrical nonlinear analysis and was obtained from Bathe et.al.[16]. The mesh for half the shell, dimensions, material properties and boundary conditions are shown in figure 8.9. The shell is treated as an axisymmetric problem. A total of 15 load increments were used and the solution was by the TS, TSV, TSG and TSVG techniques.

The assumed area of nonlinearity is shown hatched in figure 8.9 and the factors d_f=0.0, 0.1, 0.2, 0.3 and 0.4 were selected for the analysis. The results obtained are shown plotted as load factor against displacement of node 1 in figure 8.10. For d_f=0.0 the results are in

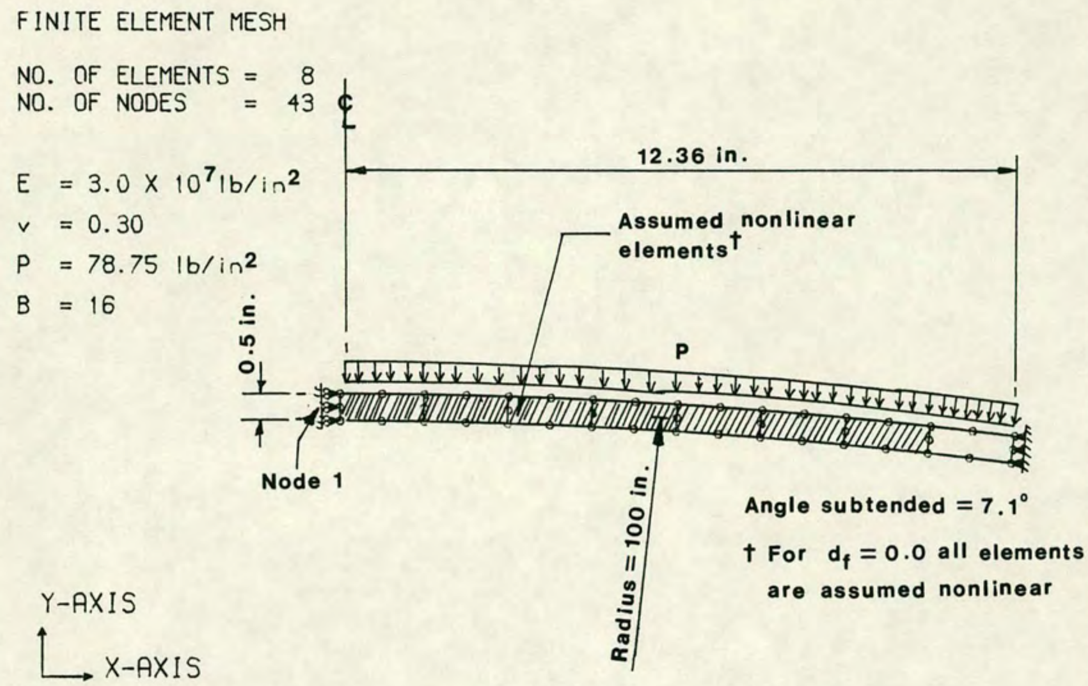


Figure 8.9

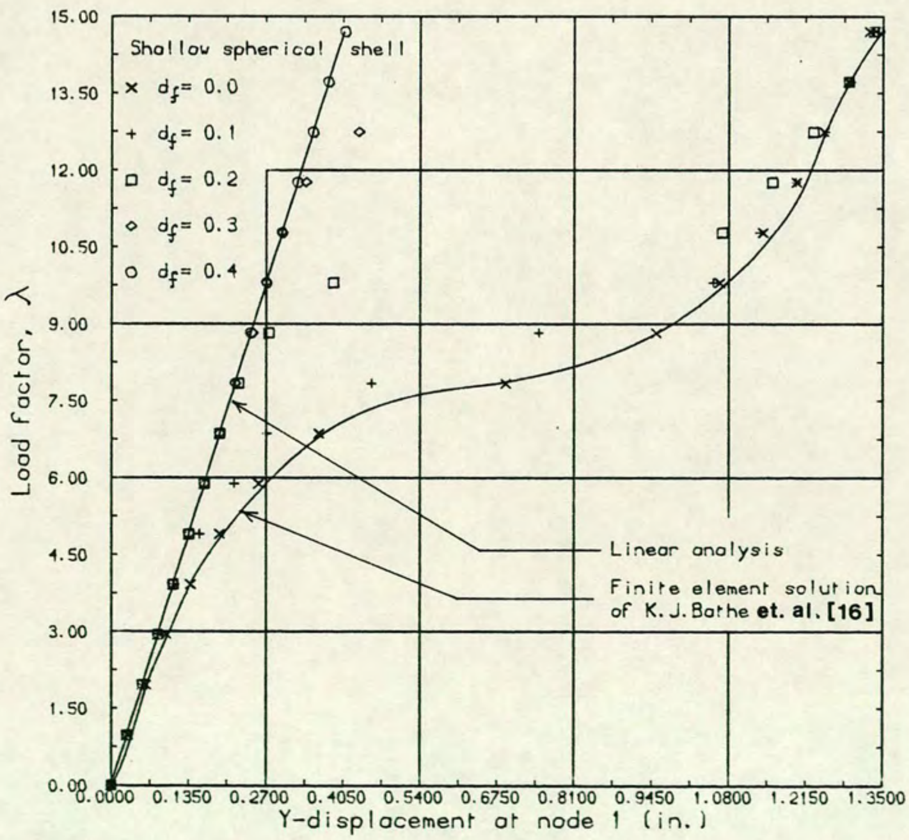


Figure 8.10

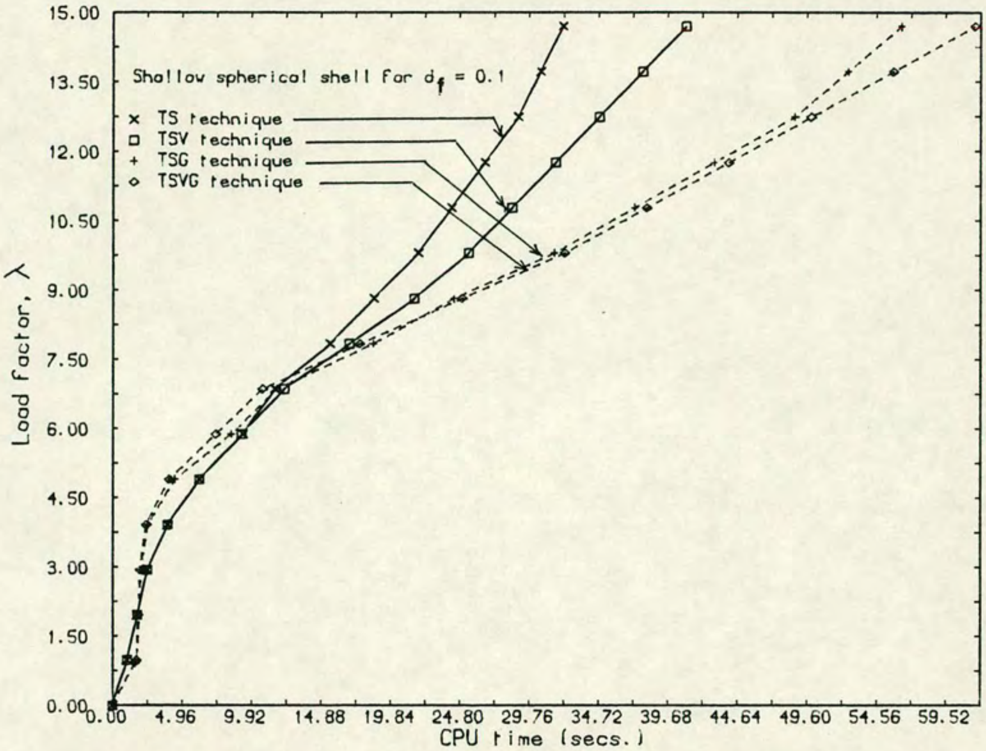


Figure 8.11(i)

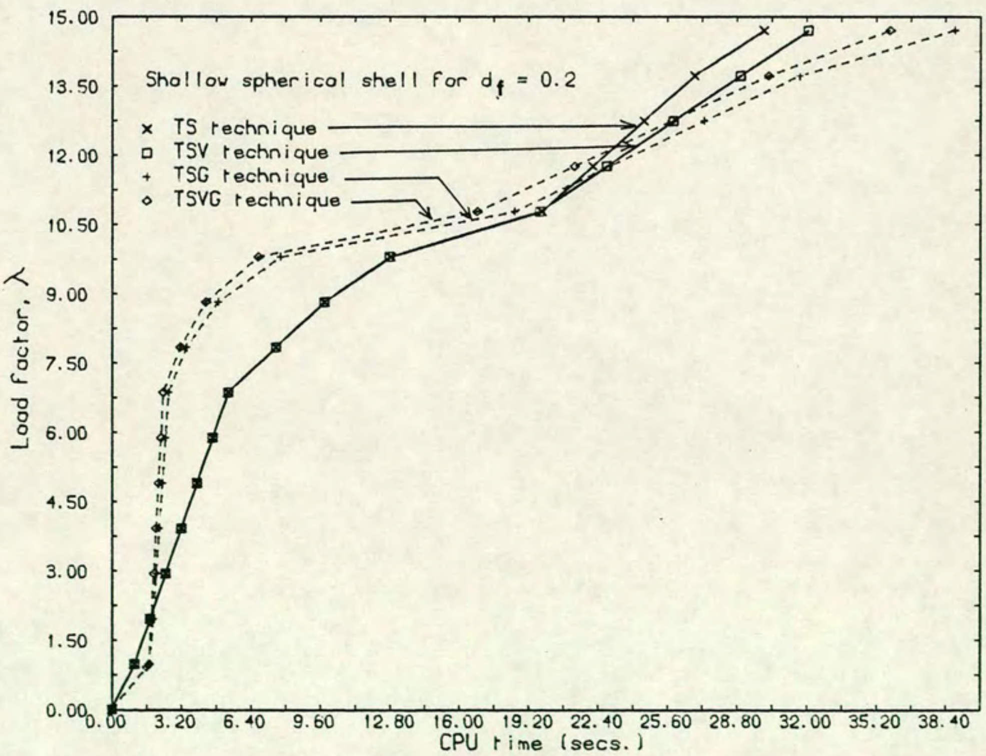


Figure 8.11(ii)

excellent agreement with those of Bathe's et.al.[16]. The structure exhibits a postbuckling behaviour where the displacements are large for a small load increment. After this the structure begins to stiffen as may be seen in figure 8.10. For larger values of d_f, the structural response is linear in the initial stages.

8.3.3.1 Graphs and table of CPU time

The graphs of load factor against the CPU time for d_f=0.1 and 0.2 are as shown in figures 8.11(i) and (ii) respectively. Table 8.4 gives the total CPU time taken for each value of d_f and the type of technique used in the analysis.

Shallow spherical shell 8 QUAD8 elements in figure 8.9 For λ= 0.0 to 14.7					
	CPU time for each value of d_f (seconds)				
Technique	d_f=0.0	d_f=0.1	d_f=0.2	d_f=0.3	d_f=0.4
TS	41.43	32.17 †	30.03 ‡	21.92	11.82
TSG	123.81	56.37 †	38.85 ‡	18.89	4.00
TSV	46.76	41.00 †	32.05 ‡	23.38	11.82
TSVG	111.96	62.00 †	36.11 ‡	19.26	3.48

Table 8.4 Total CPU time taken

† See graph in figure 8.11(i)

‡ See graph in figure 8.11(ii)

8.3.3.2 Discussion of results

For d_f=0.0, the TSG technique requires about three times the CPU time of the TS technique. For d_f=0.4, the theorems are very efficient, taking about one third of the time for the TS technique. This is because only a few elements become nonlinear at the last few load increments.

The graphs of figure 8.11(i) and (ii) clearly indicate that the theorems of geometric variation are inefficient at low factors of λ=0.0 to 2.00. This is because the initial analysis time is a large proportion of the total CPU time taken. The theorems only begin to be efficient after λ=2.00. With very large values of λ the efficiency drops as more elements become nonlinear at the later stages of the load incrementation. The results obtained for this example confirm the results of earlier examples.

8.3.4 Shallow circular arch

The nonlinear finite element analysis of this arch was studied by Bathe et.al.[16]. The mesh of 12 QUAD8 elements is shown in figure 8.12 for half the arch. The material properties, dimensions, boundary conditions and assumed area of nonlinearity are also shown in the figure. Plane stress conditions were assumed and a unit thickness was used in the analysis. A total of 19 load increments were used and the values of d_f selected were 0.0, 0.04 and 0.08.

The results are plotted in the form of load factor against displacement of node 1 in figure 8.13. As expected for d_f=0.0, excellent agreement was obtained with the solution of Bathe et.al.[16]. For values of d_f=0.04 and 0.08, the results are linear during the initial stages of loading. The analysis gradually becomes nonlinear as more elements satisfy the selected value of d_f. For such an analysis, the buckling behaviour of the arch cannot be predicted correctly. The results obtained were too 'stiff'.

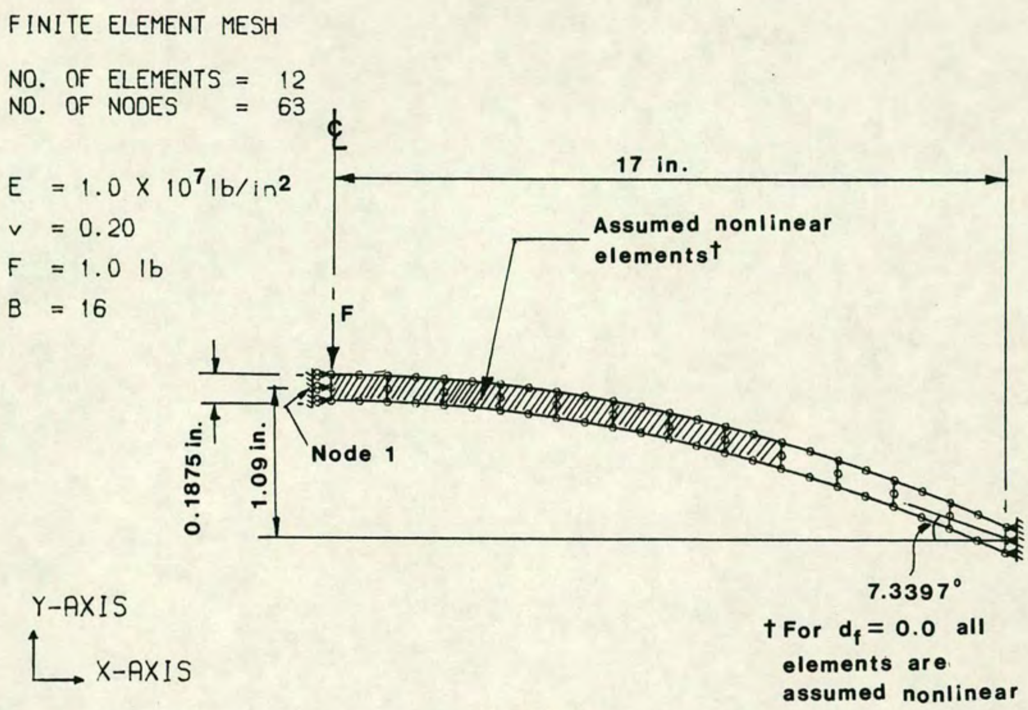


Figure 8.12

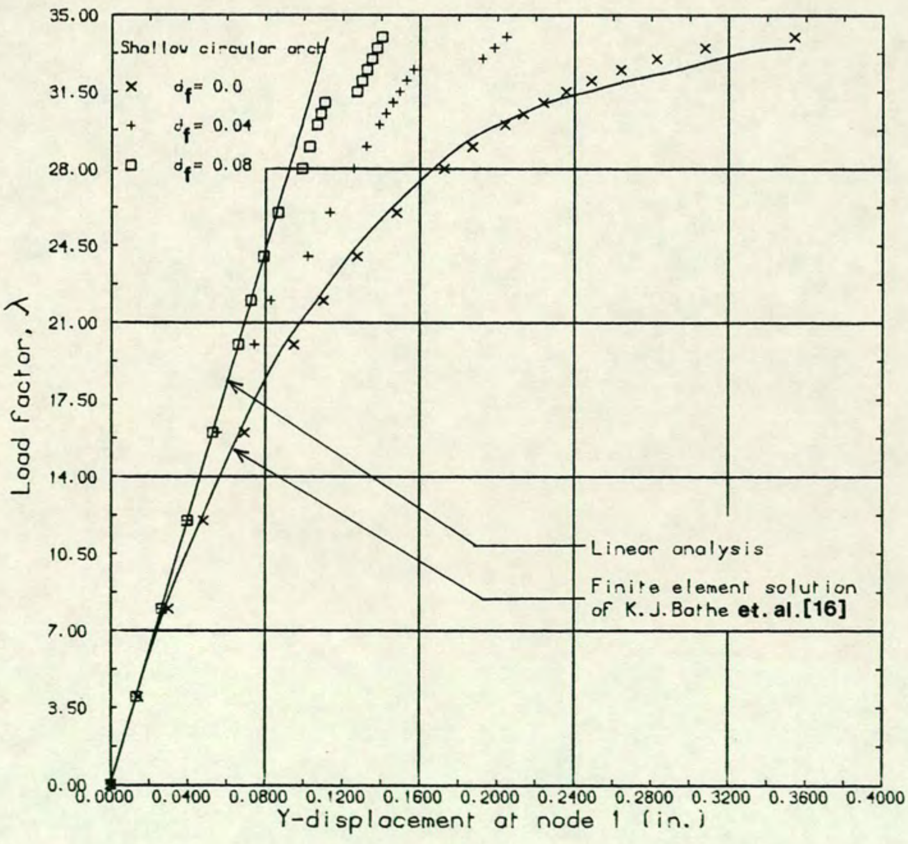


Figure 8.13

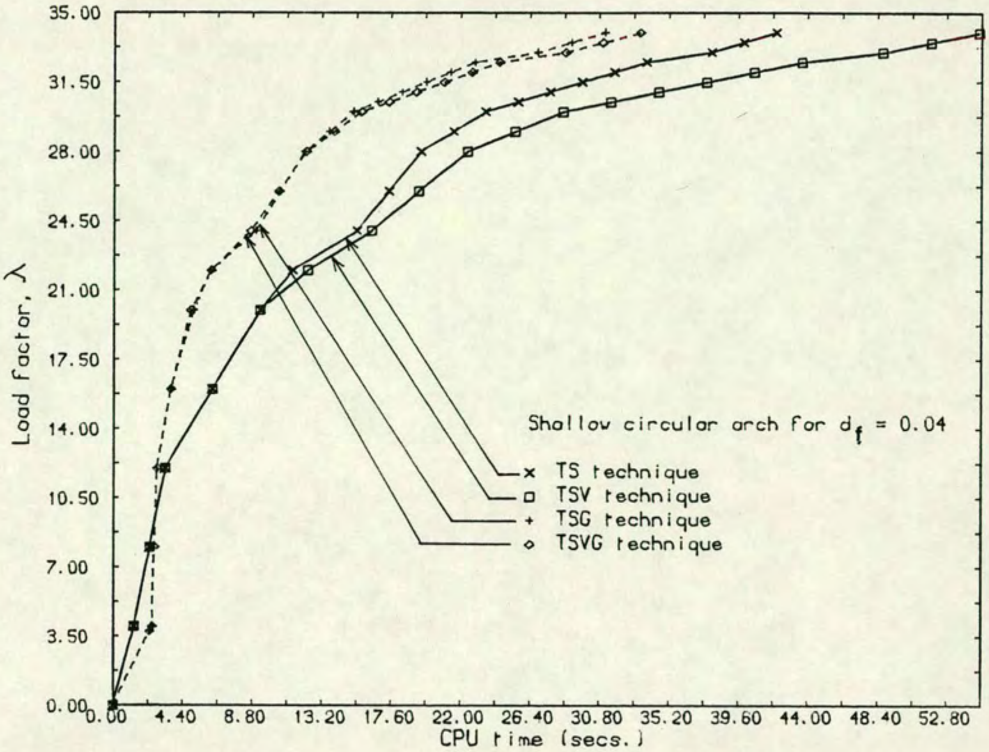


Figure 8.14(i)

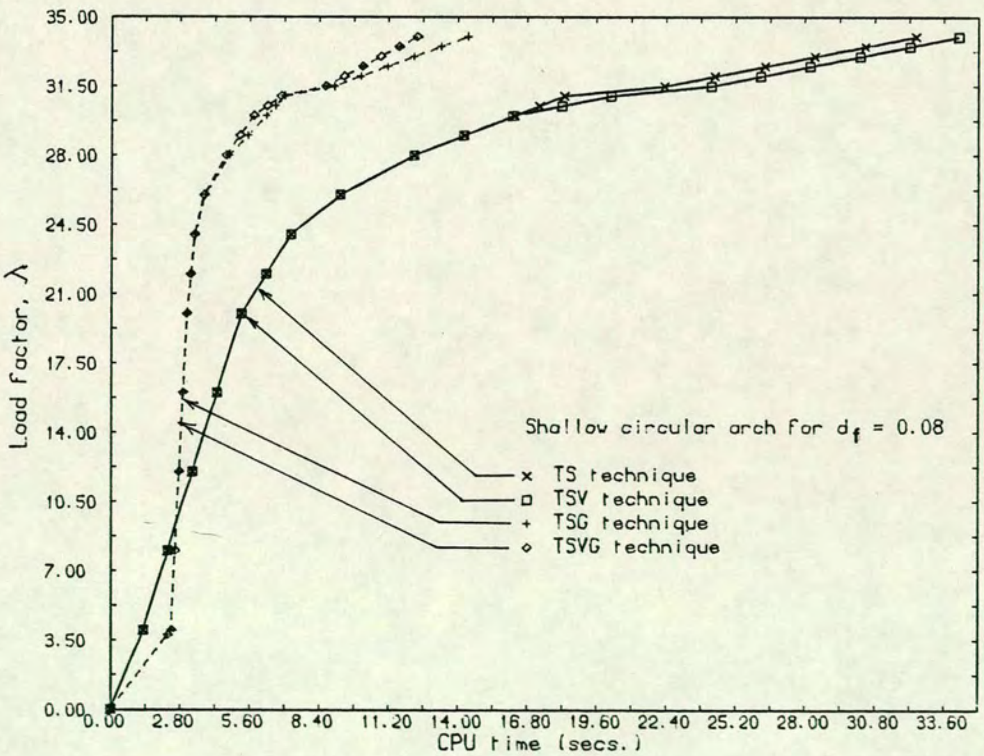


Figure 8.14(ii)

8.3.4.1 Graphs and table of CPU time

The graphs of the load factor against the CPU time during the load increments are plotted in figures 8.14(i) and (ii). The total CPU time taken for each technique and value of d_f are tabulated in table 8.5.

Shallow circular arch 12 QUAD8 elements in figure 8.12 For λ= 0.0 to 33.5			
	CPU time for each value of d_f (seconds)		
Technique	d_f =0.0	d_f =0.04	d_f =0.08
TS	51.45	42.02†	32.44‡
TSG	264.08	31.18†	14.38‡
TSV	75.82	54.82†	34.23‡
TSVG	304.12	33.40†	12.39‡

Table 8.5 Total CPU time taken

† See graph in figure 8.14(i)

‡ See graph in figure 8.14(ii)

8.3.4.2 Discussion of results

From the graphs and table, similar conclusions may be reached as given in the previous examples. The TSG and TSVG techniques are shown again to be efficient when d_f is large except during the initial stages of loading. This is because only a small number of elements are behaving nonlinearly.

8.4 Stability of the Newton-Raphson methods

The four example problems in section 8.3 were analysed by the Newton-Raphson method and its degenerate forms. The analysis was to check the stability of the IS,TS,TSV,ITS,ITSV and ITSV2 techniques for the value of d_f=0.0. The value of d_f=0.0 implies that all elements of the idealisations were assumed to behave nonlinearly. Consequently this check on the stability of the solution techniques is also applicable to the techniques using the theorems of geometric variation.

The results of the analyses are tabulated in table 8.6. The table shows which technique becomes unstable and at which load increment. Thus, for example the figure '8' in table 8.6 means that the technique is unstable at the eighth load increment. While 'full' means that the complete analysis could be undertaken up to the final load increment.

Examples of section 8.3 for d_f=0.0				
	Circular plate	Cantilever beam	Shallow shell	Shallow arch
Total no. of load increments	15	20	15	19
Technique				
IS	2	0	7	4
TS	Full	Full	Full	Full
TSV	Full	Full	Full	Full
ITS	Full	0	8	19
ITSV	Full	Full	8	Full
ITS2	Full	2	8	Full

Table 8.6 Stability of the Newton-Raphson methods

As expected the IS technique is completely unreliable as a nonlinear analysis technique for solving geometrically nonlinear problems. The ITS,ITSV and ITS2 techniques are more reliable, however it is difficult to predict which problems will these techniques be stable. The only techniques that may be relied on are the TS and TSV techniques.

8.5 Summary

The efficiency of a reanalysis technique based on the theorems of geometric variation has been investigated for the analysis of geometrical nonlinear problems. Four examples were used to indicate the particular areas where the geometric theorems may be effectively employed as well as its limitations. If an approximate solution is required, then the tangential stiffness technique by the theorems of geometric variation are competitive in comparison to the usual procedures. Furthermore the tangential stiffness technique and its variant form is generally stable for most problems. The analysis using $d_f=0.0$ is uneconomic because all elements are behaving nonlinearly.

The main application of the theorems of geometric variation will be to structures in which small discrete parts behave nonlinearly. This may occur in special and unusual structures. The proposed techniques may then be used as a preliminary investigation if such an analysis is contemplated.

CHAPTER 9

A PRELIMINARY INVESTIGATION OF THE THEOREMS OF GEOMETRIC VARIATION FOR COMBINED MATERIAL/GEOMETRICAL NONLINEAR PROBLEMS

9.1 Introduction

In this chapter the solution of combined material and geometrical nonlinear problems is investigated by the theorems of geometric variation. It has been shown that the theorems may be used successfully if the nonlinear effects are separated. Here the structural behaviour is assumed to be a combination of that described in Chapters 7 and 8.

The theorems of geometric variation are applied to some simple combined nonlinear problems. In these problems, the strains are assumed small and therefore the equations used in Chapters 7 and 8 are valid and require little modifications.

9.2 Combined material/geometrical nonlinear analysis by the theorems of geometric variation

As in the previous two chapters, the important part is the evaluation of the compensation forces. The presence of two sources of nonlinearity complicates the calculation of the elemental tangential stiffness matrix. If the strains are assumed small, then the equations of plasticity are valid for large displacements as noted in Chapter 2. This combination which is studied here, is the simplest form of combined material/geometrical nonlinearity.

The usual procedure of analysis is that all elements are assumed to behave geometrical nonlinearly with the material behaviour either elastic or elasto-plastic. The effect of the geometrical nonlinearity may either strengthen or weaken a structure which is deforming plastically. For example, if geometrical nonlinearity is included in the analysis of a plate deforming plastically, then the presence of membrane forces will strengthen the structure. In other words the plate will collapse at a higher load in comparison with the analysis of the plate where the membrane forces are ignored. However for the case of a column with an eccentric loading, the geometrical nonlinearity will weaken the structure considerably when deforming plastically. The usual procedure of analysis by the theorems will be inefficient because all elements of these structures will behave nonlinearly. The procedure to be used is therefore outlined below. The objective of this study is to establish whether the solution procedures are stable.

9.2.1 Initial analysis

For the initial analysis an assumed area of nonlinearity is required. Foreknowledge of the likely areas of geometrical, material and material/geometrical nonlinearity is therefore crucial. This may be a contentious issue, since the analysis is then heavily dependent on the ingenuity of the analyst. Nonlinear analysis will then become an art as well as a science. Problems with small discrete areas of combined nonlinear behaviour were not located by the author during his literature search. However such combined nonlinear problems may possibly occur in 'one-off' structures with special and rather unusual features. In this chapter it is assumed that the analysis of such a structure is required. The initial analysis is given by:

$$[K_o][\delta_a \mid \delta_u] = [I_a \mid I_u] \tag{9.1}$$

The initial analysis is performed at the nodes of the assumed nonlinear elements in addition to those where there are applied loadings.

9.2.2 Analysis of modified structure

The next step is to evaluate the overall matrix of compensation forces from the contribution of each nonlinear element. To do this two criteria are required to detect the presence of nonlinearities. For the material nonlinear elements this is the plastic strains and for the geometrical nonlinear elements this is the factor, d_f as defined in Chapter 8. Using these two criteria four types of elements may be identified during the analysis. These are defined below:

- **Type 1. Linear element** : This element does not satisfy the two criteria mentioned above. It is therefore not needed in the evaluation of the compensation forces.

- **Type 2. Material nonlinear element** : This element only satisfies the first criterion by the presence of plastic strains. The element deformation is small but its material behaviour is elasto-plastic.

- **Type 3. Geometrical nonlinear element** : This element only satisfies the second criterion. The deformations are assumed large but strains small and material behaviour elastic.

- **Type 4. Material/geometrical nonlinear element** : A combination of element type 2 and 3 where both criteria are satisfied. It undergoes large displacements with elasto-plastic material behaviour. The strains are assumed to be small.

Note in the usual procedure of analysis only two types of element exist. These are the type 3 and 4 elements.

The compensation forces may be evaluated using equation (4.13) for each type of element. These are given by:

For type 2 : $[C^e_{aa} \mid C^e_{au}] = - \int_{\Omega e} [B]^T [D_p][B] \, d\Omega_e \quad [\delta^e_a \mid \delta^e_u]$ (9.2a)

For type 3 : $[C^e_{aa} \mid C^e_{au}] = ([K^e_L]+[K^e_\sigma]) \quad [\delta^e_a \mid \delta^e_u]$ (9.2b)

using the elasticity matrix, [D]

For type 4 : $[C^e_{aa} \mid C^e_{au}] = ([K^e_L]+[K^e_\sigma]) \quad [\delta^e_a \mid \delta^e_u]$ (9.2c)

using the elasto-plastic matrix, $[D_{ep}]$

9.3 Investigation of the proposed techniques

Two examples were considered to investigate the proposed techniques of Chapter 4. The examples are the large displacement analysis of a simply supported elasto-plastic plate and the elasto-plastic buckling of a column.

The CPU time taken for the analysis is not given because the proposed techniques failed to converge when using the two criteria given above. By selecting the value of d_f to be some very small value, then the analysis is the usual procedure for combined material/geometrical nonlinear problems. The solution of this problem is inefficient when using the theorems since all elements are assumed to behave nonlinearly. If d_f is some large value, the analysis is simply the elasto-plastic analysis as given in Chapter 7, because the displacement criterion is never satisfied. Between these two extreme values of d_f, the theorems may be used efficiently but unfortunately the Newton-Raphson methods are unstable.

9.3.1 Simply supported circular plate

The finite element analysis of this structure was first studied by Kanchi et.al.[61]. The finite element mesh for half the plate, dimensions, material properties and boundary conditions are as shown in figure 9.1. The assumed area of nonlinearity is shown hatched in the figure. The problem was treated as axisymmetric and a total of 14 load increments were used.

Four values of d_f were selected for the analysis as 0.0, 0.15, 0.25 and 1.5. For $d_f=0.0$, the analysis is the usual procedure where all elements are behaving nonlinearly. For $d_f=1.5$, this criterion is never satisfied and the analysis becomes the small displacement elasto-plastic analysis which was used in Chapter 7. The results are plotted as load factor against displacement of node 1 in figure 9.2. Plasticity was initiated at $\lambda=2.4$ for the values of $d_f=0.0$ and 1.5. The method of solution is by the TS, TSV, TSG and TSVG techniques. For $d_f=0.0$ and 1.5 the results were in good agreement with those of Kanchi's et.al.[61].

The analysis was repeated with the value of $d_f=0.15$. At the first few load increments the analysis was linear because both criteria were never satisfied. After this some elements near the middle of the plate behave geometrical nonlinearly. However as soon as plasticity was initiated at $\lambda=2.4$ the TS, TSV, TSG and TSVG techniques begin to diverge. The other techniques of initial and initial/tangential

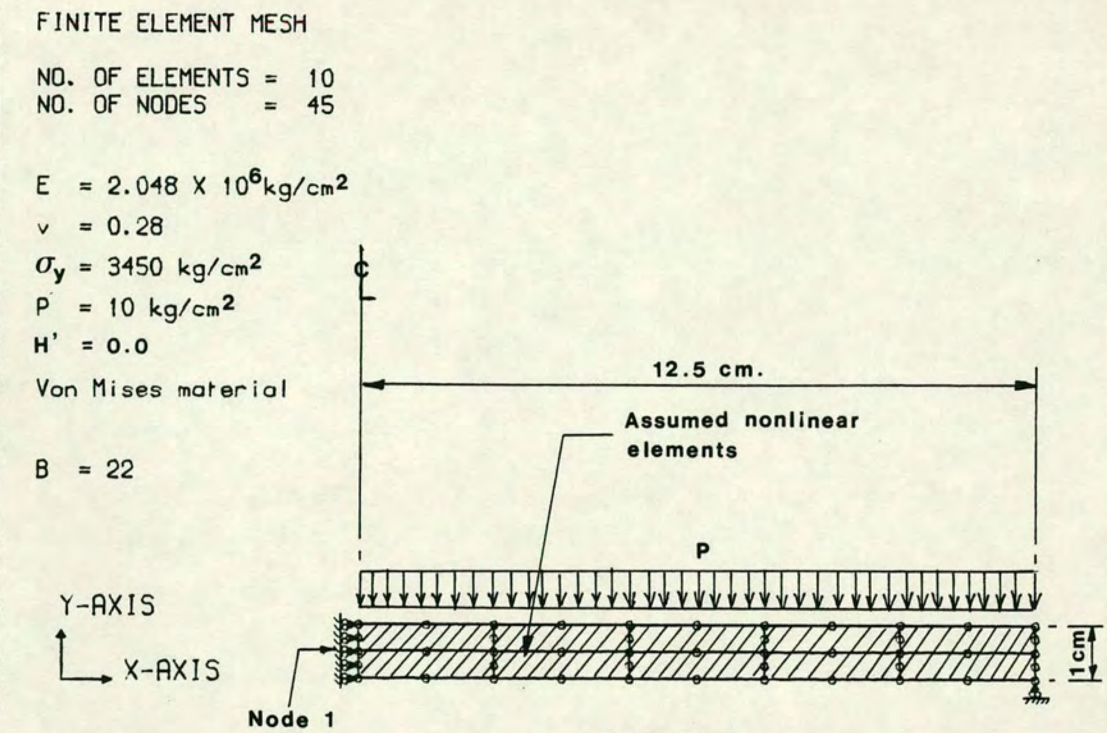


Figure 9.1

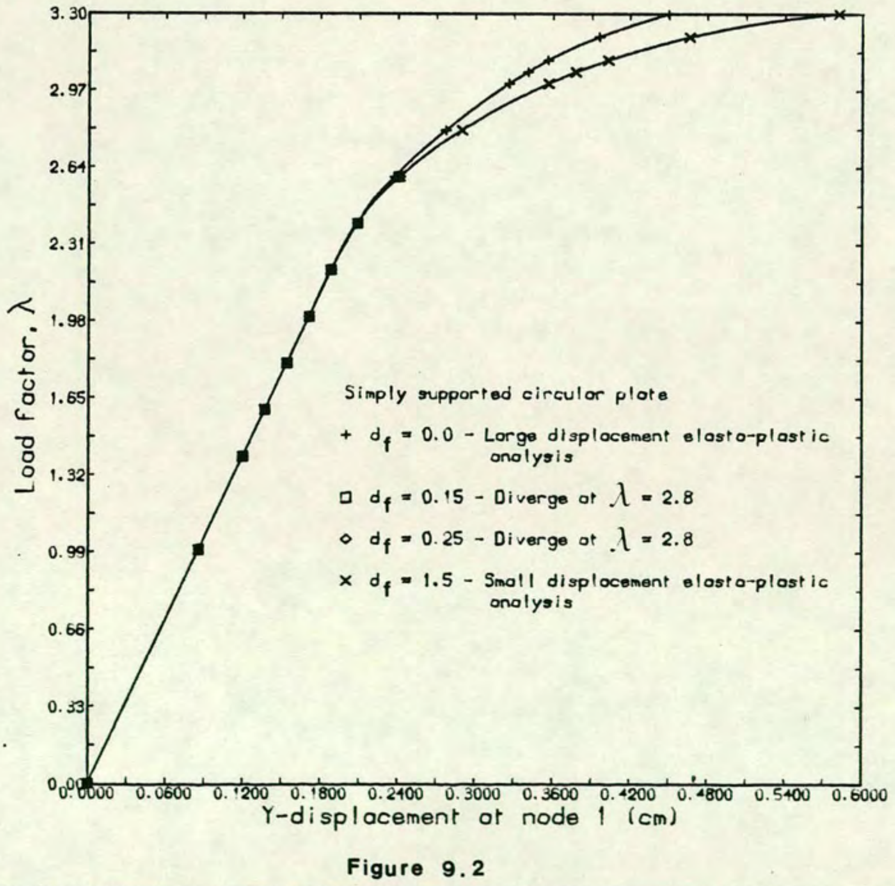


Figure 9.2

stiffness were also tried, but they also failed as soon as plasticity occurred in one of the elements. For d_f=0.25 the analysis is again linear until λ=2.4 when the plate deforms plastically. Up to λ=2.4 the analysis is identical to that for d_f=0.0. For values of λ greater than 2.4 elements near the middle of the plate behave geometrical nonlinearly and the solution immediately diverges.

For values of d_f=0.15 or 0.25, the presence of both nonlinearities occurring in some of the elements result in instability of the solution techniques. This suggests that the Newton-Raphson methods for solving such problems where four types of element may coexist is inadequate.

9.3.2 Buckling of column

This second example was also obtained from the paper by Kanchi et.al.[61]. The finite element mesh, dimensions, material properties and boundary conditions are shown in figure 9.3. Plane stress conditions and a unit thickness of the structure were assumed in the analysis. The point load was applied at an eccentricity of 0.25 m to simulate the instability effects. A total of 9 load increments were used. The assumed area of nonlinearity is shown hatched in the figure.

First a value of d_f=0.0 was selected in the analysis and the method of solution was by the TS, TSV, TSG and TSVG techniques. This is the large displacement elasto-plastic analysis and a collapse load factor of 12.7 was obtained. The results are plotted in the graph of figure 9.4. The collapse load obtained was close to the value of 12.1 quoted by Kanchi et.al.[61]. The discrepancy may be due to the different solution technique used. Kanchi et.al.[61] used a visco-plastic approach of analysis to solve this problem. When a value of d_f=0.0 was selected and σ_y was set to a large value, then the structure will never be plastic. The analysis is simply the large displacement analysis as given in Chapter 8. The results for this analysis are also plotted in figure 9.4 and are in excellent agreement with those of Kanchi's et.al.[61].

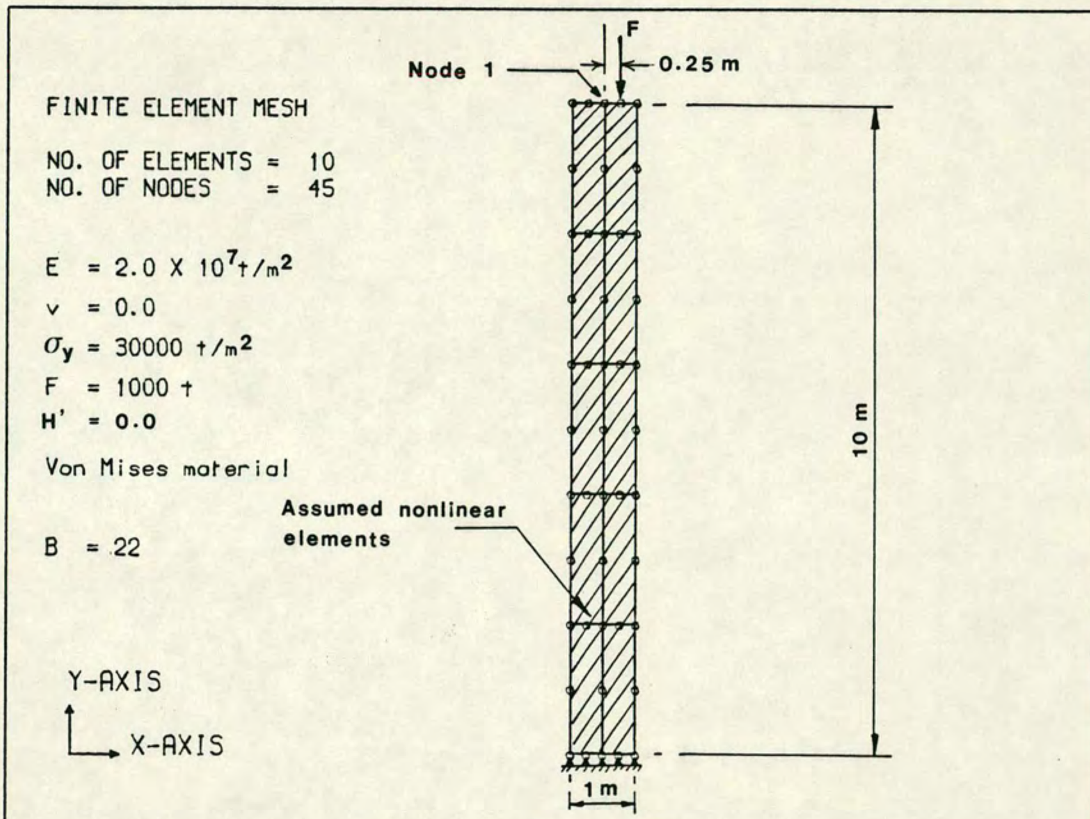


Figure 9.3

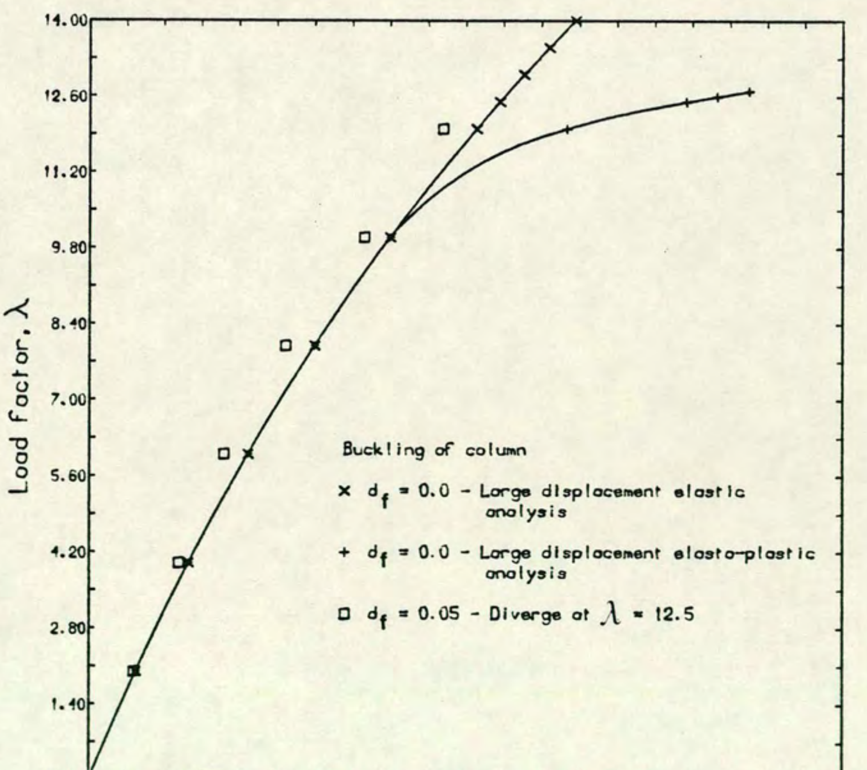


Figure 9.4

The analysis was then repeated with $d_f=0.05$ but as soon as some elements become plastic the solution diverges which occurs at $\lambda=12.5$. The other degenerate forms of the Newton-Raphson method also fail when some of the elements become plastic. As in the previous example, these techniques are inadequate to treat problems where the four types of element coexist.

An alternative approach to the problem is to assume that only two types of element may coexist during the analysis. These elements are the type 1 (linear) element and type 4 (material/geometrical nonlinear) element. The analysis was undertaken for the circular plate using the Newton-Raphson method. However, as soon as these two types of element coexist the solution immediately diverges.

9.4 Summary

The proposed techniques and the Newton-Raphson methods have been tried for material/geometrical nonlinear problems. These techniques are able to solve these problems provided that all elements in the structure behave nonlinearly. However the solution will then be inefficient by the theorems of geometric variation and therefore two criteria are needed to distinguish between linear and nonlinear elements. As a result of these two criteria more than one type of element may coexist during the analysis. The proposed techniques and the Newton-Raphson methods failed as soon as this state was attained. This was demonstrated on two examples where their existence seem to destablised the solution algorithms.

Finally, it should also be noted that such problems, may not be realistic. Examples on these type of structures have not been found in the literature. It was merely used to illustrate the limitations of the Newton-Raphson methods and the proposed techniques to treat such problems.

CHAPTER 10

GENERAL SUMMARY, CONCLUSIONS AND FUTURE RESEARCH

10.1 General summary

In the preceding nine chapters, the theorems of structural and geometric variation have been presented for linear and nonlinear finite element analysis. The theorems of structural variation were formulated in matrix form and proposed as an efficient reanalysis technique in linear finite element analysis. The efficiency of the technique was investigated to identify in which areas it may be used effectively. For the theorems of geometric variation, the formulation in matrix form was proposed as a reanalysis technique for linear and nonlinear finite element analysis. The efficiency of the reanalysis techniques using the theorems of structural and geometric variation were extensively investigated in this thesis.

The matrix formulations of the theorems of structural and geometric variation are particularly well suited for computer implementation. This is discussed in Appendix VI where the proposed techniques were developed into separate subprograms and implemented with existing computer codes. The computer programs developed were used to study the feasibility of the proposed techniques as well as to obtain solutions for simple yet practical problems.

10.2 Conclusions

This thesis has achieved the objectives outlined in the first chapter. Undoubtly several advantages and limitations of the proposed techniques have already become apparent. Nevertheless, the main conclusions of this thesis together with a brief evaluation are outlined as follows:

1. The matrix formulation of the theorems of structural and geometric variation for linear and nonlinear analysis of finite element problems are essential for efficient computer implementation. The technique of forming the overall equilibrium equations involving the modified or affected elements from each changed element is

systematic and similar to that of forming the overall stiffness matrix.

2. The theorems of structural variation were shown to be particularly efficient for locally modified large structures when a sequence of linear reanalysis is required. The form of the theorems where the condensed variation factors are evaluated directly is general and the most efficient. Two examples were presented to illustrate the potential use of the theorems for reanalysis.

3. The theorems of geometric variation are also efficient for linear reanalysis of locally modified large structures, as was proved by considering two examples. In addition the use of triangular elements should be avoided when the geometry of the structure is varied. This is because the finite element mesh becomes severely distorted and the solutions obtained are inaccurate.

4. The similarities between the theorems of structural and geometric are also shown, where the equilibrium equations of the modified elements are simply factored unbalanced forces. However the theorems of geometric variation are more general because they can deal with more complex modified structural variation involving material, geometrical and applied load changes. The modified element stiffness matrix must be formed again as it is not simply a scalar multiple of the original stiffness. This formation of the element stiffness matrix is undertaken at an extra expense.

5. The theorems of structural variation were abandoned as a nonlinear solution technique for continuum problems as explained in Chapter 4. The theorems of geometric variation on the otherhand may easily be formulated as a nonlinear solution technique based on the Newton-Raphson methods. A further technique called the variant form was introduced using the theorems of geometric variation.

6. The feasibility of the proposed nonlinear techniques were studied for material nonlinear problems. The solutions obtained were identical with that of the Newton-Raphson methods. By comparing the CPU times taken, the tangential stiffness technique using the theorems of geometric variation was shown to be the most efficient. This particular technique is suited for large structures with small areas of plasticity. those

7. In geometrical nonlinear problems, a criterion called the displacement factor was devised to detect linear and nonlinear elements. This criterion is specified by the analyst and type of analysis required. The smaller the value of this criterion the more inefficient are the proposed techniques. This is because many elements are assumed to behave nonlinearly. The proposed techniques are only efficient when the displacement factor is large but the solutions obtained only approximate to the true nonlinear behaviour.

8. For combined material/geometrical nonlinear problems two criteria are needed to identify the linear and nonlinear elements. These two criteria are the presence of plastic strains in an element and the displacement factor. However the use of these two criteria, result in four types of elements to coexist which tend to destablised the Newton-Raphson methods. It should be noted that when all elements are nonlinear, the Newton-Raphson methods are stable and the reanalysis may be undertaken using the theorems of geometric variation successfully.

9. In the computer implementation, the proposed techniques for linear and nonlinear analysis may be coded into existing computer programs without any difficulty. The task is easier if the existing programs are highly structured and divided into subprograms or modules. This is the usual case in commercial computer programs.

10. The theorems of structural and geometric variation have

been applied to a wide range of problems in linear and nonlinear analysis. These problems serve to illustrate the practical aspects, versatility and efficiency of both theorems.

11. Finally the main disadvantage of the proposed techniques is that foreknowledge of the modified areas are required if it is to be used efficiently. The techniques also require more storage space than the usual procedures. This is because the unit loads are reduced simultaneously and backing store is not used.

10.3 Future research

Although this thesis has covered a large area in studying the feasibility of the theorems of structural and geometric variation, there are still some further refinements needed. These are outlined below for future research and investigation.

1. In material nonlinear problems, it was assumed that plasticity is associative. This results in a symmetric tangential stiffness matrix. For non-associated plasticity, the stiffness matrix is unsymmetric and this requires twice as much effort to reduce the matrix by the usual Newton-Raphson methods. However the theorems of geometric variation require the reduction of the $[C_{aa}]$ matrix which is only dependent on the number of nonlinear elements. The $[C_{aa}]$ matrix does not depend on the bandwidth of the unsymmetric stiffness matrix. Hence it is expected that the theorems of geometric variation will be efficient in non-associative plasticity. The degree of efficiency of the various proposed techniques need to be investigated for such problems.

2. A further investigation of the proposed techniques for combined material/geometrical nonlinear problems. The technique should be tried using different nonlinear

solution techniques.

3. The formulation of the theorems of geometric variation based on other nonlinear solution techniques. Some of these algorithms are discussed in the survey of references [135] and [136].

4. The use of the theorems of structural variation for other nonlinear material problems. For example, the variable moduli model[28] where the modified element stiffness matrix is a scalar multiple of the original stiffness is suitable. This model was used by Duncan and Chang[35] for finite element analysis of soil problems.

5. The use of the proposed nonlinear techniques in dynamic analysis. In dynamic analysis many reanalysis are required during a time interval.

6. A comparison with the other reanalysis techniques that were reviewed in Appendix I. This should be based on efficiency, versatility of applications and ease of computer implementation.

7. In this thesis the band solver has been used for the reduction of the multiple load cases. Comparison of efficiency may be investigated with the frontal and skyline solution techniques which are popular in finite element analysis.

APPENDIX I

A REVIEW OF STATIC REANALYSIS TECHNIQUES

I.1 Introduction

In structural design or optimization the procedures are generally iterative and require repeated analysis, as the structure is progressively modified. In order to avoid a fresh analysis after each iteration, many reanalysis techniques have been devised. These reanalysis techniques were reviewed by Arora[10] in 1976. This review will obviously contain some duplications but papers that had been overlooked by Arora[10] and new techniques are discussed. Arora[10] gives a concise definition of the problem when he states that it is "to find the response of a structure after modifications using the original response of the structure such that the computational time of reanalysis is less than the complete analysis time". At the same time reanalysis techniques allow one to efficiently compute the design sensitivity coefficients required for optimization methods. Reanalysis techniques are particularly important for large structures, especially in finite element idealisations where only a small part of the structure is progressively modified. Such reanalysis techniques may be easily implemented in computer codes without any difficulty.

Unfortunately texts specially devoted to reanalysis techniques and their computer implementation are not readily available. At most only a chapter on these neglected techniques is included in numerical texts. For example; Atrek et.al.[13 Chapter 17], Fox[40 Chapter 5], Kirsch[68 Chapter 5], Majid[82 Chapter 6], Meek[89 Chapter 6], Pestel and Leckie[113 Chapter 9,10] and Pipes[116 Chapter 6]. Therefore there is a need to consider the various aspects of these techniques.

Reanalysis techniques may be broadly classified as either direct (i.e. exact) or iterative and approximate methods. A table[137,161] showing the relationship between most of these methods is given on figure I.1. These techniques are either formulated using the force (or flexibility) method, stiffness (or displacement) method or mixed methods of analysis. Judicious selection of a technique is important to ensure that the method may be easily applied and is also efficient.

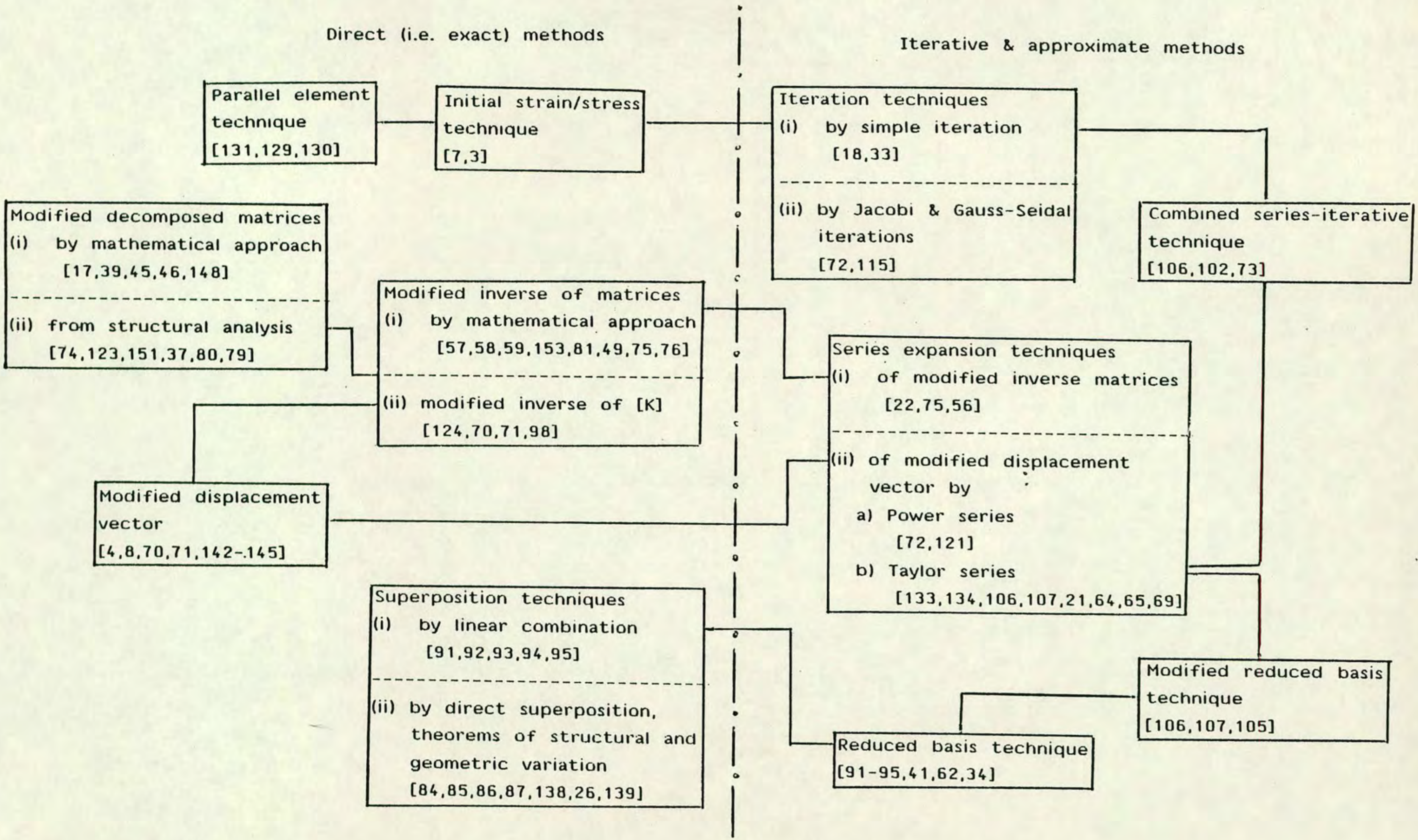


Figure I.1 Reanalysis techniques

I.2 Direct (i.e. exact) methods

These methods give an exact closed-form solutions which have the same effect of solving afresh the modified system of equations. In general direct methods are efficient if the number of modified elements are small.

I.2.1 Initial strain technique

The method of 'initial strain or stress' was first formulated in matrix notation by Argyris and Kelsey[7,3] in 1956. The earlier reference [7] uses the force method of analysis because it was felt then, that it was superior compared with the stiffness method. However the subsequent paper[3] gives the formulation for both the force and stiffness method. This reanalysis technique was used for the analysis of structures with cut-outs (the total removal of elements) and modifications of elements.

The technique was initially developed to avoid the difficulties of analysing structures with cut-outs by the force method. Instead the results of the complete structure were used to predict the response of the modified structure. The matrices needed in the force method for a complete structure are much more easier to form than a structure with cut-outs. With the force method, the original structure is analysed for the external loadings and initial strains are imposed on the elements to be removed such that their stresses due to the external loadings and initial strains are reduced to zero. The magnitude of the initial strains are unknowns but these may be found from their corresponding stresses acting on the original structure due to external loadings only. The modified stresses are found in terms of the original stresses. The technique requires only one analysis of the original structure. For the stiffness method initial unknown element stresses are imposed in addition to the applied loads. This is the dual approach of the force method.

The technique had already been developed as far back as in 1945 by Best[20,19] and was exploited by Cicala[29] and Michelson and Dijk[97]. These early methods were suitable for the repeated analysis of structures by hand. Cicala[29] used special perturbation stress systems to nullify the stresses in the cut-outs. The perturbation was then added to the stresses obtained for the complete structure. Michelson and Dijk[97] presented the technique to account for variations in cross-sectional areas and elastic modulus. Goodey[48] proposed the same technique but the formulation used a variational argument (the minimisation of the strain energy). This approach was a purely mathematical concept to obtain the desired modifications. The matrix form of Argyris and Kelsey[7,3] was used by Poppleton[117,118] for the redesign of redundant aircraft structures by the force method. Here the inverse problem was posed; the stresses were specified and the required changes in the structure were determined for a given load system. This was similar to the method illustrated in the examples given by Best[20,19], and results in redesign problems that require iterations to ensure that the specified stresses were not violated.

The initial strain concept was disputed by Grzedzielski[50] who introduced fictitious thermal loads to replace the initial strains. In addition he suggested that the method was limited for analysing structures with a purely diagonal flexibility matrix. In a series of articles by Grzedzielski, Argyris and Kelsey[6,51] the validity of the initial strain concept was discussed.

The initial strain or stress techniques require the inverse of the original flexibility or stiffness matrix respectively. The efficiency of the technique for the stiffness method was investigated by Kavlie and Powell[63,149]. Using a system of operation counts, they concluded that this reanalysis technique is inefficient when compared with a fresh analysis. In addition an error in the original derivation was corrected by the authors.

I.2.2 Parallel element technique

In 1968 Sobieszczanski[131] introduced the parallel element concept in matrix form using the force method of analysis. This concept is a perturbation technique which originated from the earlier work of Cicala[29] and Michelson and Dijk[97]. It was indirectly suggested in the series of papers by Grzedzielski, Argyris and Kelsey[50,6,51] in their controversy over the initial strain concept.

The technique uses superposition of an element parallel to that of the one to be modified. It may be used for the addition (where none had existed), deletion and modification of elements and hence was more general than that of the initial strain concept. The differences between this method and the initial strain concept were also explained. Here the original structure is updated after each modification; the results of the previous modifications form the basis of the response of the new structure. Whereas with the initial strain concept, subsequent analyses were always referred to the original structural analysis. The parallel element technique was subsequently formulated for the displacement method by Sobieszczanski[129,130]. These papers include tests on efficiency and accuracy of the parallel element concept compared with that of the initial strain concept. The efficiency was measured in terms of operation counts and computer time. The author concluded that the technique is very efficient for a large number of modifications. The accuracy of the technique also appeared not to be affected by the magnitude of the modifications. Errors that were obtained were assumed to be random and did not accumulate. At the same time a version of the technique using the displacement method was recognised to be more efficient than that of the force technique[130]. This is because of the sparsity of the stiffness matrix compared to that of the flexibility matrix.

I.2.3 Modified inverse of matrices

The two previous techniques were based on intuitive and physical reasoning rather than a mathematical concept. Many investigators have sought the relationship between the change in stiffness, the inverse of the original and modified stiffness matrices. One approach is to express the modified inverse explicitly in terms of the original inverse and the changes in the stiffness.

The required relationship may be based on the Sherman-Morrison identity[128,127]. Householder[57,58 Chapter 2, 59 Chapter 5] provided an equation between these three quantities in matrix form. Zielke[153] also derived a similar equation for the inverse of modified symmetric matrices. The modified inverse may be built up a row or a column at a time in any order, and it is also possible to proceed coefficient by coefficient. It was subsequently used by Sack et.al.[124] where the modified inverse was obtained one column at a time. Using operation counts the technique was shown to be efficient in comparison to Gaussian elimination when the modifications are small. Kavlie and Powell[63,149] have also presented an operation count of Sack's[124] technique. They found that it is inefficient compared to a complete reanalysis using the stiffness method.

Kirsch and Rubinstein[70,71] also investigated various versions of the Sherman-Morrison identity. These versions involve forming the modified inverse by considering either; simultaneous changes of the coefficients or changing the coefficients (or columns) one at a time. The most efficient way involves changes of the columns.

Another variation of the Sherman-Morrison identity was given by Mohraz and Wright[98], where the size of the stiffness matrix was allowed to change. For addition of nodes the size of the modified inverse increases and for deletion it decreases. Using operation counts their proposed method suggested savings of 20%-80% in computational effort compared to a complete inversion.

Other similar methods due to MacNeal[81] and Goodey[49] were presented in the 1950's. These methods use the original inverse matrix to obtain the modified inverse matrix. MacNeal's[81] method is analogous to the compensation theorem used in network theory and was somewhat simplified by Kosko[75]. Later Kosko[76] derived various techniques for inversion of matrices (including for modified matrices) by partitioning. The possibility of sparsity of a matrix common in the displacement method was not considered. This was probably due to the popularity of the force method at that time.

I.2.4 Modified displacement vector

The technique of calculating the modified inverse matrix directly is expensive. An alternative approach is to calculate the modified displacement vector using the Sherman-Morrison identity[128,127]. This approach avoids the necessity of forming the inverse explicitly.

The technique of Sack et.al.[124] has several drawbacks as pointed out by Argyris et.al.[4]. The main objection was that to compute the inverse of a large and banded stiffness matrix is expensive, and that to store it may at times be uneconomical. Secondly the Sherman-Morrison identity considers modification of one column at a time. Based on these objections Argyris et.al.[4] produced yet another algorithm to be used with the displacement method. Instead of the modified inverse, the modified displacement vector was calculated by rearranging the Sherman-Morrison identity. The method makes use of the sparsity and bandwidth of the stiffness matrix during the reduction process by Cholesky decomposition. It is particularly efficient if changes occur at the higher numbered nodes of the structure, i.e. 'near the bottom' of the stiffness matrix. This technique was shown using operation counts[4] to be more efficient than that of Sack et.al.[124].

Kirsch and Rubinstein[70,71] also used the Sherman-Morrison identity where the generation of the modified inverse was avoided. Two versions were presented to calculate the modified displacement vector. One of these versions which they called the 'method of reduced equations' was the most efficient compared to the techniques of the previous section.

Kavlie and Powell[63,149] also presented a similar technique to calculate the modified displacement vector. Their technique was shown to be inefficient compared to a complete reanalysis except for very small modifications. However for cases when a linear search is being made in an optimization algorithm, the technique will become very competitive.

As a result of earlier work[4] Argyris and Roy[8] proposed a general method of reanalysis where the sparsity and banded nature of the stiffness matrix is taken into account. The proposed method involves changes in the size of the stiffness matrix, much in the same way as Mohraz and Wright[98]. However it is more efficient as the inverse is not required. The method is extended for updating the decomposed stiffness matrix after each modification. It may also be used in substructures where elements within it are modified. This technique is applied to problems of crack extension and closure by Armen[9]. Its implementation in a general purpose computer program was undertaken by Raibstein et.al.[119]. Here the efficiency was also investigated for large and complex structures that occur in the design environment. The authors showed that for an efficient reanalysis the maximum percentage of the degrees of freedom that may be modified using this method varies between 8%-60%.

A development that has recently appeared in the literature, by Wang et.al.[142,143,144,145] is similar to that of Argyris and Roy[8]. The modified response of a structure is expressed as a linear combination of the original response and a term depending on the pseudo-loads. These pseudo-loads are related to changes in the stiffness of the original system and analysed with the applied loads. A reduced set of equations are set up for the modified parts of the structure only. They called this technique the 'pseudo-load method'. An application of the pseudo-load method with static condensation in finite element analysis was given by Hirai et.al.[55]. The technique was used for the reanalysis of a fine mesh using the results of a coarse mesh.

I.2.5 Modified decomposed matrices

This technique has received particular attention in recent years. This is because of the frequently used finite element displacement method results in a large symmetric and banded stiffness matrix. The matrix is decomposed and the displacements obtained by back-substitution. For minor modifications, it is more efficient to update the decomposed matrix rather than form and reduce the new stiffness matrix.

Bennett[17] recognised that in the case of a sparse and banded matrix, calculating the complete inverse may be avoided. Instead, he proposed an algorithm that enabled the modified decomposed matrix to be obtained from the original decomposed matrix.

Here the modified triangular factors of a matrix are computed from the original triangular factors. Argyris et.al.[4,8] also considered this technique as part of their general modification method discussed in the previous section. Variants of Bennett's[17] technique have also appeared in the mathematical literature[39,45,46]. An earlier paper by Weiner[148], which appeared in 1948, considered a similar problem. In this paper the solution procedure was tabulated for hand calculation and checking purposes. More importantly in the ensuing discussions, it was pointed out that the technique is suitable for studying structures of similar configurations with different structural properties. However its advantage in not using the inverse matrix was not recognised.

An application of Bennett's[17] algorithm to finite element analysis was derived by Young[152]. To use the technique the local nonlinear behaviour must be expressed in a particular form. A revised version of the technique for nonsymmetric changes in an initially symmetric matrix was given by Kleiber and Lutoborski[74]. Other similar techniques of updating the decomposed matrix were given by Row et.al.[123] and Yang[151]. Row et.al.[123] developed two algorithms using the Crout and Cholesky methods to decompose the matrix. Using operation counts and CPU time, the Cholesky method was shown to be more efficient for obtaining the modified decomposed matrix.

Ertas and Fenves[37] have also introduced three different reanalysis techniques; two of which are based on the stiffness method and the other on the force method of analysis. The first two techniques were the 'modified stiffness method' which is similar to those of Kosko's[75] and MacNeal's[81], and the 'modified Gauss method' which is similar to that of Bennett's[17] algorithm. The last one was the 'modified flexibility method' which is similar to that of Argyris and Kelsey's[7,3] initial strain concept. The efficiency of these three techniques were compared and the modified Gauss was shown to be the most efficient. However this was for the case when the modifications were restricted to a predefined region of interest.

The main disadvantage of the previous techniques in this section is that for efficiency the modifications should be near the bottom of the matrix. This requires foreknowledge of the changes and possibly a rearrangement of the stiffness coefficients in the matrix. To circumvent these difficulties a recent technique has been proposed by Law and Fenves[80,79]. Their approach is to consider matrix modification problems, sparse matrix methods, substructure analysis and graph theory together. Various improvements to the algorithms by Bennett[17], Row et.al.[123] and Yang[151] were given. In addition an algorithm was suggested where the coefficients of the original matrix need not be stored as required in the previous techniques. As the decomposed matrix is known, then to obtain the original coefficients a reverse process of the reduction was used. The use of substructures in their strategy suggest that modifications may be randomly distributed without prior knowledge of the structure. Unfortunately their algorithms rely heavily on graph theory which is an unfamiliar topic for many engineers. Similar ideas on sparse matrix techniques for reanalysis were also given by Lam et.al.[78].

I.2.6 Superposition techniques

Melosh et.al.[91,92,93,94,95,38] introduced a superposition technique similar to the initial strain or stress concept. One was based on the complementary energy approach where the modified forces were expressed as a linear combination of the original forces and self-equilibrating force vectors. The other used the potential energy approach with the original displacements and self-straining vectors to obtain the modified displacements. The self-equilibrating forces or strains may be selected to give the exact result, but approximate procedures are also possible. The response of the modified structure is obtained by superimposing the response of the initial structure and the response due to self-equilibrating force vectors. Kavlie and Powell[63,149] have given an assessment of the efficiency of this technique using count operations. For one load case a complete reanalysis is more efficient. For five load cases and when the number of elements to be modified was greater than two, the technique was still inefficient for their example problem.

Another more recent technique by Majid and co-workers[84,85,86,87] was introduced in the 1970's. This technique was based on the superposition of unit and applied load analyses. It was called the 'theorems of structural variation' and used in the optimum design of pin and rigid-jointed structures[85,86]. These theorems were first able to handle modification of one element at a time. Bakry[14] has shown that simultaneous modification of two or more elements may be undertaken. Extension to finite element problems was undertaken by Topping[138] and its simplification by Atrek[12]. Its application in nonlinear analysis for framed structures is shown by Celik and Majid[24,25,83]. However the method only caters for changes in structural properties like cross-sectional areas, thickness or moment of inertia. The extension of these theorems to account for changes in structural geometry was only achieved recently by Topping et.al.[26,139]. These new theorems are hence called the 'theorems of geometric variation' and may also include the effect of changes in structural properties. Although unit load analyses are required, these theorems are attractive because the design sensitivity coefficients are obtained for use in structural optimization. However efficiency studies (based on CPU times or operation counts) have not been carried out.

I.3 Iterative and approximate methods

Approximate methods are generally derived from some form of a series expansion. An early review of approximate methods was given by Schmit[125] in 1971. Iterative methods apply successive corrections to the initial solution and converge to a more accurate solution for the modified structure. In these methods solution accuracy and rate of convergence are important. Computational effort and efficiency has to offset against accuracy. These two factors can rarely be simultaneously satisfied, because a more refined solution decreases the efficiency and vice versa. Therefore some compromise must be made such that the 'run-time' is not excessive and that the accuracy of the solution is sufficient. The optimum balance between these two opposing factors depends on how critical is the accuracy for a particular design problem. Usually these methods would be used to evaluate various design alternatives quickly. For the initial and final designs, an exact analysis would be carried out.

I.3.1 Iteration techniques

In 1963 Best[18] presented a reanalysis technique using simple iteration. He called the technique the 'equivalent load method'. The equivalent load is calculated from ({F}-[ΔK]{δ}) where: {F} is the vector of applied loads; [ΔK] is the change in stiffness matrix; and {δ} is the vector of the current value of displacements at each iteration. The reduced form of the stiffness matrix is available from the first iteration and hence only reduction of the equivalent loads is required. The improved solution is then obtained by back-substitution. This simple iteration technique is very similar to the initial stiffness method used in nonlinear analysis. When [ΔK] is small, convergence is rapid but for large changes it may be slow or diverge. In Best's[18] method the total displacements are calculated after each iteration. A slightly different form is given by Das[33] where the incremental displacements are calculated instead. Here the equivalent load is given by -[ΔK]{δ} at each iteration. The numerical results obtained by Das[33] indicate that three iterations are required for most structural problems provided that changes in stiffness are kept within 10% of the original values. Substantial savings in computer time may be made as shown by Das[33]. The author also suggested that the error increases with more severe changes. This suggests that the simple iteration technique is only useful for small changes in stiffness where convergence is rapid and errors small.

Kavlie and Powell[63,149] counted the number of operations per iteration required for the simple iteration technique. For small changes and few load cases the technique was shown to be efficient when compared with a fresh analysis. For large changes the technique may in some cases not converged at all. In addition the authors suggested that convergence may be improved for large changes if an under-relaxation factor is used.

In order to improve the convergence of the simple iteration technique, Kirsch and Rubinstein[72] introduced another technique. They expressed the change in stiffness, [ΔK] as a linear combination of two matrices. The relationship that was obtained was shown to be that of the Jacobi iteration. A further improved technique was suggested where the iteration for each component of $\{\delta\}$ was performed instead of the whole vector. Each technique was compared for changes in stiffness of 100% or more. The simple iteration technique fails for such large changes although it requires less operations. The two proposed techniques require slightly more operations but this was shown to be more than offset by the improved convergence. The improved technique of calculating each component of $\{\delta\}$ was proved to be superior.

Phansalkar[115] considered splitting the stiffness matrix in different ways and arrived at the simple, Jacobi and Gauss-Seidal iterations. In the Gauss-Seidal iteration, components of $\{\delta\}$ for each iteration are successively used to compute the remaining components. The question of convergence and effectiveness of the various iterative schemes were also considered. One such scheme that was found to be efficient is the Block Gauss-Seidal iteration. This technique involves adjustment of groups of unknowns as opposed to the point Gauss-Seidal iteration where only one unknown is adjusted at a time. It was further suggested that acceleration methods may be used and easily implemented to improve the efficiency of this technique.

I.3.2 Series expansion techniques

An early example of a series expansion technique was presented by Brock[22]. With this technique the modified inverse matrix was obtained by an infinite series expansion. An exact relationship was derived by summation of the series provided that the changes are small. Kosko[75] also considered an infinite series to obtain the modified inverse. A similar approach by Hoerner[56], uses only the first term of this series. The technique was shown to converge after one iteration for changes in the stiffness of less than 35%. For large changes (by a factor of 3.6 of the original stiffness) it converges after four iterations. This technique is no longer a serious contender because as mentioned earlier, it is inefficient to work with inverses.

Alternatively the approximate modified displacement vector may be derived from various series expansions. Kirsch and Rubinstein[72] for example, used the binomial expansion to derive their Jacobi iteration technique. Romstad et.al.[121] considered a general power series expansion to obtain any desired degree of accuracy for changes in the stiffness. For static analysis their power series expansion was the same as the binomial expansion. This technique was investigated by Arora and Rim[11] and found to be unsuitable even for small changes in stiffness. Zimmermann and Spence[160] also used the power series expansion to study the effect of changing one element. The change in one element affects a group of elements by a global parameter. This effect was termed 'tracking sensitivity' and was subsequently used as an algorithm for interactive finite element analysis.

Storaasli and Sobieszczanski[133,134] used the first-order Taylor series expansion for reanalysis of large complex structures. This technique requires calculation of sensitivity coefficients which was then used to find a better approximation of the displacements of the modified structure. These sensitivity coefficients may be obtained by decomposing a pseudo-load term and then back-substituting using the original decomposed stiffness matrix. They investigated the efficiency and accuracy of this technique for modification of one element (for changes of -100% to 500%) and more then one element (for changes of -50% to 50%). The results indicated that the errors in displacement and stresses are less then 16%. The time taken was only a fraction of that for a complete analysis.

Noor and Lowder[106] investigated various reanalysis techniques. The Taylor series expansion technique was shown to be inaccurate after one iteration for changes in stiffness greater than 20%. The error can be reduced after two iteration cycles. In a later paper[107] they considered the first and second-order Taylor series expansions for the mixed method of analysis. The second-order Taylor expansion was the more accurate technique but at the expense of more computational and storage requirements. The accuracy of the first-order Taylor expansion method was shown to improve if reciprocals of the design variables (which are the cross-sectional areas, thickness etc.) were used. The use of reciprocals was also favoured by Schmit and Farshi[126]. In the first-order Taylor expansion of the displacement and mixed method of reanalysis, the displacements obtained were identical but the forces from the mixed method were more accurate.

Bhatia[21] presented a reanalysis technique where the stiffness matrix was first reduced by static condensation. The choice of which degrees of freedom to retain depends on the type of problem. It was suggested that to improve the efficiency the condensed matrix should be transformed to a generalised matrix of lower order by the normal mode method of structural dynamics. The generalised matrix should then be expanded by using the Taylor series about the original structure. However the formulation was not tested for any numerical example.

Recently Kirsch[64,65,69] presented a Taylor series expansion for design variables in the displacement method and their reciprocals in the force method. The expansion was shown to be equivalent to a series from simple iteration. The Taylor series expansion was then used to formulate for reanalysis along a line (i.e. one variable). This particular form of reanalysis was used with optimization techniques. The changes in the design variables may be expressed in terms of a single independent variable. A polynomial and a modified nonpolynomial approximation were derived using the Taylor expansion in terms of the single independent variable. The case studies indicate that the use of reciprocals provided better results when using the polynomial approximation than simply using the design variables. The nonpolynomial approximation gave results that were closed to the exact solution, even when the behaviour is sensitive to changes. Since only a single independent variable is involved in the reanalysis, both techniques require less computations than the usual procedure. To improve the approximations, dynamic acceleration and acceleration techniques involving scaling of the design variables were also introduced. Other approximations were also suggested using quadratic and cubic interpolations. This technique requires two exact analysis and better approximations were obtained but at extra computations. All these techniques were placed in the context of optimization design by Kirsch[67,66].

I.3.3 Combined series - iterative technique

To overcome the disadvantages of the simple iterative and first-order Taylor series techniques, Noor and Lowder[106] suggested that a combination of both would be profitable. The first approximation was obtained from the Taylor series which was then used as an estimate for the simple iterative technique. In this combined technique, one iteration cycle can significantly improve the accuracy and efficiency of the approximation. When compared with the modified reduced basis technique (see section I.3.5) it leads to the same accuracy for changes less than 20%.

This combined technique was then used by Noor[102] in the mixed method of analysis for modifications of the structural geometry. By comparison with the Taylor series expansion technique, the combined technique was proved to be superior.

A similar combined technique was proposed by Kirsch and Toledano[73], for changes in the geometry of the structure. The new procedure is based on combining simple iteration and scaling of the original structural analysis. The simple iteration part was expressed as a series expansion. To improve the quality of this approximation, scaling of the initial design was introduced. It was concluded that the new proposed technique was adequate in terms of accuracy but involves more calculations per iteration compared to other simpler techniques.

I.3.4 Reduced basis techniques

One possible approximation technique is based on the reduced basis idea. Here the response of the modified structure is expressed as a linear combination of known independent vectors. The number of these vectors is less than the number of structural degrees of freedom. In other words the modified behaviour of the structure is approximated using fewer degrees of freedom. Melosh and Luik[91,92,93,94,95,38] proposed two techniques for how these vectors are selected. The selection is as outlined in the section on exact methods (I.2.6). This approximate technique requires less operations for several reanalysis cycles[63,149]. This is in contrast to the version of the technique for an exact analysis.

Fox and Miura[41] presented a similar technique where the modified displacements were expressed as a linear combination of previously computed displacement vectors. These displacement vectors were obtained from previous changes of the structure. The changes were the basic design vectors. The choice of these basic designs appears to be on an ad hoc basis. It was also suggested that the technique is equivalent to applying a Ritz-Galerkin principle. This particular form was shown to be efficient when compared with the computational effort for a new analysis.

A similar approach was introduced by Kavanagh[62]. The displacement vector was expressed as a linear combination of the eigenvectors of the original structure. The choice of eigenvectors depended on their energy contribution; only those with significant contributions were included in the approximation. The basis of the technique is the normal mode method of structural dynamics. The changes in the structure were introduced as a nonlinearity into the normal mode equations and solved by dynamic relaxation. The technique performs best for global changes rather than local ones.

The application of this technique for nonlinear analysis by Noor and Peters[109,108,103] has recently appeared. The independent vectors are those used for the static perturbation technique.

More recently Ding and Gallagher[34] introduced a reduced basis formula using the force method of analysis. The formula may be derived to give an exact or approximate response of the modified structure. The redundant forces were selected as the reduced basis since they were fewer than the total number of forces in the structure. Their case studies showed that the exact and in particular the approximate techniques are efficient. The authors concluded that both these techniques were effective and that even the results of the approximate method were sufficiently accurate for the purpose of redesign during an optimization process. However the techniques are limited to the reanalysis of frame and truss structures, and work to increase its versatility is in progress.

I.3.5 Modified reduced basis technique

A modified reduced basis technique was developed by Noor and Lowder[106]. This technique is a combination of the first-order Taylor series and the reduced basis methods. The choice of the independent vectors were those of the original solution and the first-order sensitivity coefficients. Some numerical studies and results were given which showed that the modified reduced basis technique is highly accurate for a wide range of modifications. Furthermore the authors showed that the choice of the independent vectors is rational. Noor and Lowder[107] concluded that this technique and the first-order Taylor series expansion with the mixed method of analysis offers the highest potential in terms of accuracy and efficiency.

The technique was further developed for use in substructuring[105]. The main difference with this development was that the original solution was not included as the independent vector. The design variables and their reciprocals would give the same results. This version of the technique was shown to be accurate and efficient for an analysis of a large structure.

I.4 Final comments

There are numerous reanalysis techniques as shown in the review. The choice of the technique will depend on the type and size of the problem, the number and magnitude of modifications, efficiency and accuracy required. There is no superior technique best suited for all problems. Comparisons between various techniques based on 'standard tests' are limited[11,63,149,72,73,106,107]. In general direct methods are applicable to situations where a relatively small portion of the structure is modified. Iterative and approximate methods are more efficient, but the accuracy of the solution in some cases may not be sufficient. Experience with the various methods is perhaps the best guide to the appropriate choice and their applications.

It should be remembered that for computer implementation that there is no need to program exactly in the same way as in the derivations. The derivations are merely used for conforming to matrix formalism and should not necessarily be carried out in actual programming. This in many cases would lead to unnecessary large number of operations and a waste of computer time. Where possible the use of sparsity of the matrices would result in reduction in computer time and storage requirements. Therefore it is essential to use the optimum sequence of operations and data storage.

The main advantages of using a reanalysis technique are:

1. Many redesign cycles in structural optimization may be undertaken at a relatively low cost.

2. It is possible to treat nonlinearity efficiently by solving a reduced set of equations.

3. The use of interactive analysis becomes possible for the design of large structures.

APPENDIX II

SOME BASIC EQUATIONS FROM THE THEORY OF ELASTICITY

II.1 Introduction

The derivation of the element stiffness matrix is facilitated by some knowledge of elasticity. The equations that are presented here are for homogeneous and isotropic materials. For a detail treatment and proofs of these equations see Washizu[146]. All the equations are for a general three-dimensional body. The application to plane stress, plane strain and axisymmetric solids may be found in references [53,54,112,154]. The equations given here are the necessary ones for the finite element method.

II.2 Strain-displacement equations

The relations between the components of strain and displacement, $\{\delta\}=\{u,v,w\}$ at a point $\{X\}=\{x,y,z\}$ are:

$$\varepsilon_x = \frac{\partial u}{\partial x} + \frac{1}{2}\left[\left(\frac{\partial u}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial x}\right)^2 + \left(\frac{\partial w}{\partial x}\right)^2\right] \tag{II.1a}$$

$$\varepsilon_y = \frac{\partial v}{\partial y} + \frac{1}{2}\left[\left(\frac{\partial u}{\partial y}\right)^2 + \left(\frac{\partial v}{\partial y}\right)^2 + \left(\frac{\partial w}{\partial y}\right)^2\right] \tag{II.1b}$$

$$\varepsilon_z = \frac{\partial w}{\partial z} + \frac{1}{2}\left[\left(\frac{\partial u}{\partial z}\right)^2 + \left(\frac{\partial v}{\partial z}\right)^2 + \left(\frac{\partial w}{\partial z}\right)^2\right] \tag{II.1c}$$

$$\gamma_{xy} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} + \frac{\partial u \partial u}{\partial x \partial y} + \frac{\partial v \partial v}{\partial x \partial y} + \frac{\partial w \partial w}{\partial x \partial y} \tag{II.1d}$$

$$\gamma_{yz} = \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} + \frac{\partial u \partial u}{\partial y \partial z} + \frac{\partial v \partial v}{\partial y \partial z} + \frac{\partial w \partial w}{\partial y \partial z} \tag{II.1e}$$

$$\gamma_{zx} = \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} + \frac{\partial u \partial u}{\partial z \partial x} + \frac{\partial v \partial v}{\partial z \partial x} + \frac{\partial w \partial w}{\partial z \partial x} \tag{II.1f}$$

Equation (II.1) is valid whether displacements or strains are large or small[154]. The strain vector $\{\varepsilon_x,\varepsilon_y,\varepsilon_z,\gamma_{xy},\gamma_{yz},\gamma_{zx}\}^T$, is known as Green's strain and used in Chapter 2 for the derivation of geometrical nonlinearity. For linear analysis only the first order terms of equation (II.1) are retained:

$$\varepsilon_x = \frac{\partial u}{\partial x} \quad ; \quad \varepsilon_y = \frac{\partial v}{\partial y} \quad ; \quad \varepsilon_z = \frac{\partial w}{\partial z} \tag{II.2a}$$

$$\gamma_{xy} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \quad ; \quad \gamma_{yz} = \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \quad ; \quad \gamma_{zx} = \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \tag{II.2b}$$

These are the engineering strains used for the linear analysis in Chapter 2.

Equations (II.2) and (II.1) are used to form the strain displacement matrix, [B] for linear and nonlinear analysis respectively.

II.3 Stress-strain equations

For a homogeneous and isotropic material only two independent constants are needed to define the elasticity matrix. This is given as:

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{bmatrix} = \Theta_1 \begin{bmatrix} 1 & \Theta_2 & \Theta_2 & 0 & 0 & 0 \\ & 1 & \Theta_2 & 0 & 0 & 0 \\ & & 1 & 0 & 0 & 0 \\ & & & \Theta_3 & 0 & 0 \\ & \text{SYMMETRIC} & & & \Theta_3 & 0 \\ & & & & & \Theta_3 \end{bmatrix} \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{bmatrix} \tag{II.3a}$$

where:

$$\Theta_1 = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \tag{II.3b}$$

$$\Theta_2 = \frac{\nu}{1-\nu} \tag{II.3c}$$

$$\Theta_3 = \frac{(1-2\nu)}{2(1-\nu)} \tag{II.3d}$$

$\{\sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{yz}, \tau_{zx}\}^T$ = stress vector;

E = Elastic modulus; and

ν = Poisson's ratio.

If the strains of equation (II.2) are used with equation (II.3), the stresses are the engineering stress. The engineering strain and stress are used for linear finite element analysis derived in Chapter 2.

The use of Green's strain in equation (II.1) with equation (II.3) results in stresses known as the 2nd. Piola-Kirchoff stresses. It is assumed that the elasticity matrix is the same for linear and geometrical nonlinear analysis. These strain and stress measures are used for the nonlinear finite element analysis in Chapter 2.

APPENDIX III

THE NATURAL STIFFNESS TRIANGULAR ELEMENT

III.1 Introduction

The natural stiffness element developed by Argyris[2,5] is expressed by a (3 by 3) stiffness matrix instead of the usual (6 by 6). Here the nodal forces and displacements, are expressed as edge forces and extensions. The edge forces and extensions act along the sides of the element. This element is used as an analogy with truss members to derive the theorems of geometric variation for finite element problems in Chapter 3. It is also used in Chapter 5 to investigate the efficiency of the theorems of structural variation.

As this is not a familiar element that is normally used in finite element problems, the element stiffness are derived in this Appendix. The derivation follows directly from Topping[137].

III.2 Derivation of element stiffness

A 3-node triangular element with nodal and edge forces is as shown in figure III.1.

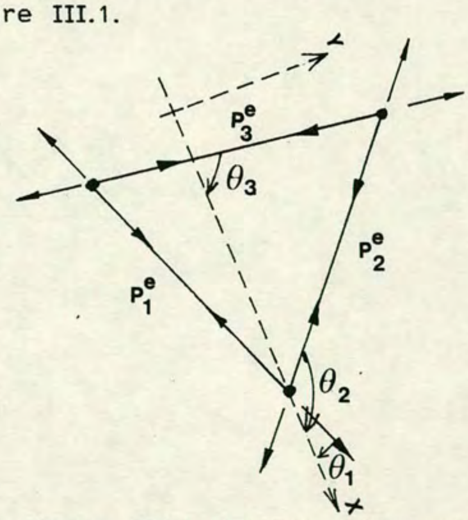


Figure III.1 Natural stiffness element

The edge forces corresponding to the edge extensions are:

$$\{P^e\} = \begin{bmatrix} P_1^e \\ P_2^e \\ P_3^e \end{bmatrix} \tag{III.1}$$

The subscripts 1,2 and 3 denote sides 1,2 and 3 respectively.

The state of stress of the element is defined by the edge extensions of the triangle instead of the nodes. The edge strains of the element are defined as:

$$\{e\} = \begin{bmatrix} e_1 \\ e_2 \\ e_3 \end{bmatrix} \tag{III.2}$$

The edge strains, {e} are evaluated from the edge extensions, $\{d^e\}$ and the side lengths:

$$\begin{bmatrix} e_1 \\ e_2 \\ e_3 \end{bmatrix} = \begin{bmatrix} 1/L_1 & 0 & 0 \\ 0 & 1/L_2 & 0 \\ 0 & 0 & 1/L_3 \end{bmatrix} \begin{bmatrix} d_1^e \\ d_2^e \\ d_3^e \end{bmatrix} \tag{III.3a}$$

or

$$\{e\} = \lceil L \rfloor \{d^e\} \tag{III.3b}$$

where:

L_i = side length;

d_i^e = edge extension; and

i = 1,2,3.

The edge strains, {e} may be expressed in terms of the element strains, $\{\varepsilon\}$ by:

$$e_i = \varepsilon_x \cos^2\theta_i + \varepsilon_y \sin^2\theta_i + \gamma_{xy} \cos\theta_i \sin\theta_i \tag{III.4}$$

where the angle θ_i are as shown in figure III.1. By rearranging equation (III.4) the element strains may be expressed in terms of the edge strains. This is given by:

$$\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} = 1/D \begin{bmatrix} (b_2c_3 - b_3c_2) & -(b_1c_3 - c_1b_3) & (b_1c_2 - b_2c_1) \\ -(a_2c_3 - a_3c_2) & (a_1c_3 - a_3c_1) & -(a_1c_2 - a_2c_1) \\ (a_2b_3 - a_3b_2) & -(a_1b_3 - a_3b_1) & (a_1b_2 - a_2b_1) \end{bmatrix} \begin{bmatrix} e_1 \\ e_2 \\ e_3 \end{bmatrix} \qquad \text{(III.5a)}$$

or

$$\{\varepsilon\} = [G]\{e\} \qquad \text{(III.5b)}$$

where:

$$D = a_1(b_2c_3 - b_3c_2) - a_2(b_1c_3 - b_3c_1) + a_3(b_1c_2 - b_2c_1) \qquad \text{(III.5c)}$$

$$a_i = \cos^2\theta_i \qquad \text{(III.5d)}$$

$$b_i = \sin^2\theta_i \qquad \text{(III.5e)}$$

$$c_i = \sin\theta_i\cos\theta_i \qquad \text{(III.5f)}$$

The element stresses are related to the element strains by the elasticity matrix:

$$\{\sigma\} = [D]\{\varepsilon\} \qquad \text{(III.6)}$$

The element stiffness matrix may be derived using the virtual work principle. From equation (2.3) of Chapter 2 (for one element):

$$\{d_*^e\}^T\{P^e\} = \int_{\Omega_e}\{\varepsilon_*^e\}^T\{\sigma^e\}\, d\Omega_e \qquad \text{(III.7)}$$

In equation (III.7) virtual nodal displacements are replaced by virtual edge extensions. Similarly the nodal forces are replaced by the edge forces to give the external virtual work on the left side of equation (III.7).

Substituting equation (III.3b) into (III.5b), the virtual strains are expressed in terms of the virtual edge extensions as:

$$\{\varepsilon_*^e\} = [G]\lceil L\rfloor\{d_*^e\} \qquad \text{(III.8)}$$

Substituting equations (III.3b) and (III.5b) into (III.6), the element stresses are given by:

$$\{\sigma\} = [D][G]\{e\} = [D][G]\lceil L\rfloor\{d^e\} \qquad \text{(III.9)}$$

Equations (III.8) and (III.9) are substituted into the virtual work equation (III.7), and by imposing unit virtual edge extensions gives:

$$\{P^e\} = \int_{\Omega_e} ([G]\lceil L \rfloor)^T [D]([G]\lceil L \rfloor)\, d\Omega_e \;\; \{d^e\} \qquad \text{(III.10a)}$$

or

$$\{P^e\} = \int_{\Omega_e} [B]^T [D][B]\, d\Omega_e \;\; \{d^e\} \qquad \text{(III.10b)}$$

and the natural stiffness element is given by:

$$[K_N^e] = \int_{\Omega_e} [B]^T [D][B]\, d\Omega_e \qquad \text{(III.10c)}$$

where:

$$[B] = [G]\lceil L \rfloor \qquad \text{(III.10d)}$$

The resulting stiffness matrix, $[K_N^e]$ will be (3 by 3).

In Chapter 3 equation (III.10a) was used to calculate the edge forces. The edge extensions are first evaluated from the nodal displacements and by using the natural stiffness element, $[K_N^e]$ the edge forces are obtained.

In Chapter 5 the natural stiffness element was used to investigate the efficiency of the theorems of structural variation. Pairs of equal and opposite unit loads are applied along the edges of the element. The resulting edge forces from equation (III.10a) are used to form the equilibrium equations of the modified element.

APPENDIX IV

AN EXAMPLE USING ONE-DIMENSIONAL FINITE ELEMENTS

IV.1 Introduction

The formulations for the theorems of structural variation given in Chapters 3 and 4 may be applied to the nonlinear analysis of finite elements. To illustrate each of the algorithms a simple one-dimensional finite element problem capable of hand solution has been considered. The two node one-dimensional constant stress finite element is basically the same as an axially loaded pin-jointed member. The stiffness matrix of the element, $[K^e]$ may be derived by the usual finite element procedure using linear interpolation functions[112]:

$$[K^e] = \frac{AE_T}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \tag{IV.1}$$

where:

A = cross-sectional area of the element;

L = length of the element; and

E_T = tangent modulus of the material obtained from the uniaxial stress-strain curve at stress, σ.

The stress-strain curve of the material may be specified by a relationship of the form:

$$\sigma = f(\varepsilon) \quad \text{or} \quad \varepsilon = g(\sigma) \tag{IV.2a,IV.2b}$$

where f and g are some functions of the strain and stress respectively. From the equations above the tangent modulus is defined as:

$$E_T = \frac{d\sigma}{d\varepsilon} = f'(\varepsilon) \quad \text{or} \quad E_T = \frac{d\sigma}{d\varepsilon} = \frac{1}{g'(\sigma)} \tag{IV.3a,IV.3b}$$

For simplicity of the hand calculations the expression used for the stress-strain relationship for the nonlinear elements as suggested by Owen and Hinton[112] was:

$$\sigma = E_o(\varepsilon - 5\varepsilon^2) \qquad \text{(IV.4)}$$

where E_o is some reference value of material modulus.

Hence:

$$E_T = \frac{d\sigma}{d\varepsilon} = E_o(1 - 10\varepsilon) \qquad \text{(IV.5)}$$

The strain of the element is given by:

$$\varepsilon = (\delta_2 - \delta_1)/L \qquad \text{(IV.6)}$$

where δ_1, δ_2 are the displacements at nodes 1 and 2 of the element. Tensile strains are defined as positive and compressive strains as negative; similarly for the stresses. The equivalent nodal forces corresponding to the element stresses are f_1 and f_2 are given by:

$$f_1 = -f_2 = \sigma A \qquad \text{(IV.7)}$$

IV.2 Initial analysis

The problem to be studied is shown in figure IV.1(i) with the applied loading in figure IV.1(ii). The behaviour of the first element is assumed nonlinear as given by equations (IV.4) and (IV.5) and the second element is assumed linear. Both elements have section properties of A = 1.0 and E_o = 200.0. Since the first element is nonlinear unit load analyses are only required at nodes 1 and 2. The unit load at node 1 will not effect the solution since this node is fixed. However it will be included in the formulation to illustrate the general solution of the problem. The unit load analyses that are required are as shown in figures IV.1(iii) and (iv). Assuming that the first estimate of the displacements $\{\delta^0\}=\{0\}$ the strains in the elements are given by $\varepsilon_1 = \varepsilon_2 = 0$. The stiffness of the nonlinear element is given by equations (IV.1) and (IV.5) as:

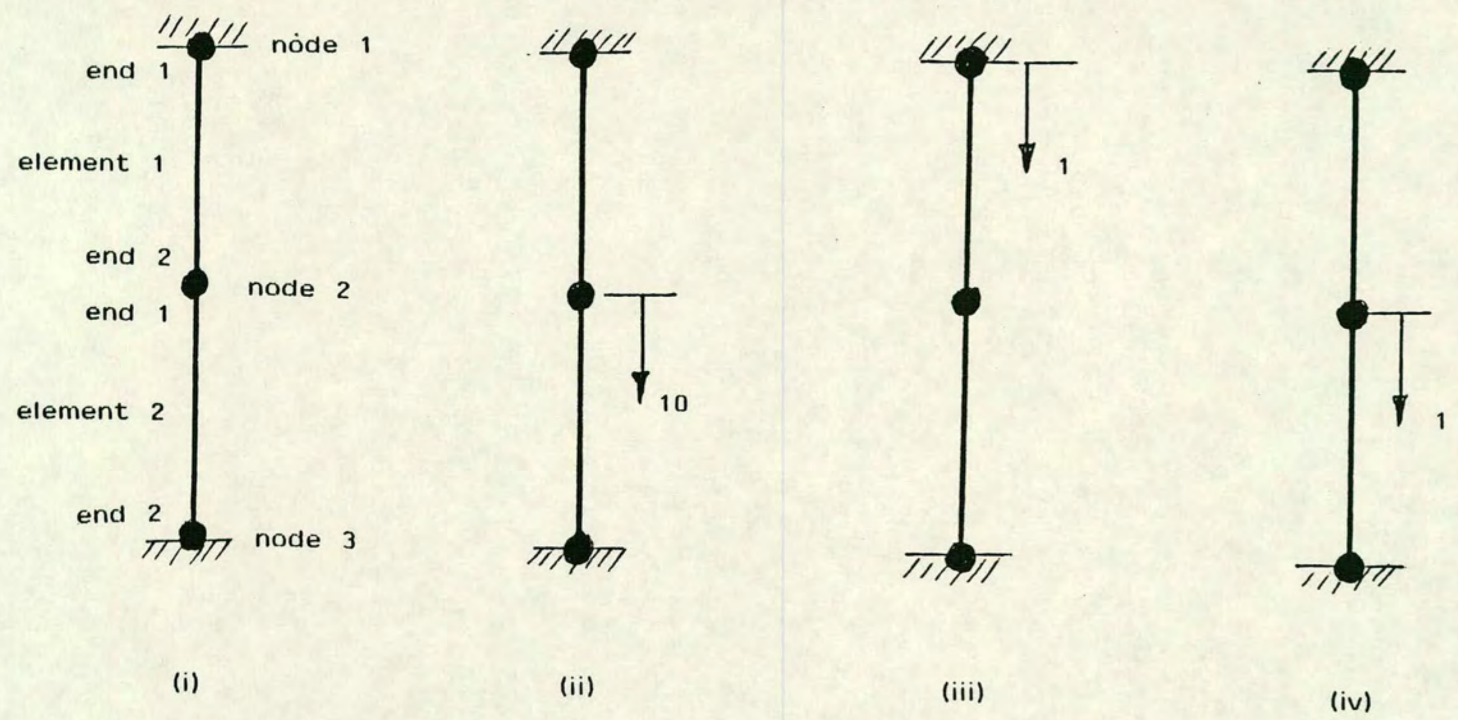

end 1
node 1
element 1
end 2
node 2
end 1
element 2
end 2
node 3
(i)
10
(ii)
1
(iii)
1
(iv)

Figure IV.1

$$[K^e] = \frac{A}{L} E_o (1-10\varepsilon) \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \begin{bmatrix} 200 & -200 \\ -200 & 200 \end{bmatrix}$$

The stiffness of the linear element is the same. Solution of the stiffness equation (4.1) under the unit and applied loads are given by:

$$\begin{bmatrix} 200 & -200 & 0 \\ -200 & 400 & -200 \\ 0 & -200 & 200 \end{bmatrix} \left[\begin{array}{cc|c} \delta_{11} & \delta_{12} & \delta_1 \\ \delta_{21} & \delta_{22} & \delta_2 \\ \delta_{31} & \delta_{32} & \delta_3 \end{array}\right] = \left[\begin{array}{cc|c} 1 & 0 & 0 \\ 0 & 1 & 10 \\ 0 & 0 & 0 \end{array}\right]$$

Applying the boundary conditions (displacements at nodes 1 and 3 are zero) and solving gives:

$$\left[\begin{array}{cc|c} \delta_{11} & \delta_{12} & \delta_1 \\ \delta_{21} & \delta_{22} & \delta_2 \\ \delta_{31} & \delta_{32} & \delta_3 \end{array}\right] = \left[\begin{array}{cc|c} 0 & 0 & 0 \\ 0 & 0.0025 & 0.025 \\ 0 & 0 & 0 \end{array}\right]$$

The internal forces of the first nonlinear element due to the unit loads are given by:

$$\begin{bmatrix} f_{11} & f_{12} \\ f_{21} & f_{22} \end{bmatrix} = \begin{bmatrix} 0 & -0.5 \\ 0 & +0.5 \end{bmatrix}$$

IV.2.1 Initial stiffness technique

The nodal displacements from the first iteration are obtained from the initial analysis as, $\{\delta^1\}=\{0,0.025,0\}$ and the element strains may be calculated using equation (IV.6) giving:

$$\varepsilon_1 = +0.025 \quad \text{and} \quad \varepsilon_2 = -0.025 .$$

The element stresses may be calculated using equation (IV.4) (for the nonlinear element only) giving:

$$\sigma_1 = +4.375 \quad \text{and} \quad \sigma_2 = -5.000$$

The equivalent nodal forces are given by equation (IV.7). These nodal forces may be calculated first using the original element properties and subsequently using the modified nonlinear element properties. Subtracting these two values gives the out of balance forces at each end of the element which may be used to calculate the residual forces at the nodes as follows:

Element	Nodal force (Original stiffness)	Nodal force (Nonlinear stiffness)	Out of balance force
1	$f_1 = -5.000$ $f_2 = +5.000$	$f_1 = -4.375$ $f_2 = +4.375$	$f_{b1} = -0.625$ $f_{b2} = +0.625$
2	$f_1 = +5.000$ $f_2 = -5.000$	$f_1 = +5.000$ $f_2 = -5.000$	$f_{b1} = 0.000$ $f_{b2} = 0.000$

The unbalanced forces of the second element are zero, since this element is linear. Using equation (4.2a) the condensed variation factors are given by:

$$\begin{bmatrix} R^2_{c1} \\ R^2_{c2} \end{bmatrix} = - \begin{bmatrix} \psi^2_1 \\ \psi^2_2 \end{bmatrix} = \begin{bmatrix} -0.625 \\ +0.625 \end{bmatrix}$$

The nodal displacements may be calculated using equation (4.2b) as follows:

$$\begin{bmatrix} \delta^2_1 \\ \delta^2_2 \\ \delta^2_3 \end{bmatrix} = \begin{bmatrix} \delta^1_1 \\ \delta^1_2 \\ \delta^1_3 \end{bmatrix} + \begin{bmatrix} \delta_{11} & \delta_{12} \\ \delta_{21} & \delta_{22} \\ \delta_{31} & \delta_{32} \end{bmatrix} \begin{bmatrix} R^2_{c1} \\ R^2_{c2} \end{bmatrix}$$

$$= \begin{bmatrix} 0.0 \\ 0.025 \\ 0.0 \end{bmatrix} + \begin{bmatrix} 0 & 0 \\ 0 & 0.0025 \\ 0 & 0 \end{bmatrix} \begin{bmatrix} -0.625 \\ +0.625 \end{bmatrix} = \begin{bmatrix} 0 \\ 0.02656 \\ 0 \end{bmatrix}$$

In the third iteration of the analysis the strains and stresses may be calculated using the results of the second analysis and equations (IV.6) and (IV.4) to give:

$\varepsilon_1 = +0.02656$ and $\varepsilon_2 = -0.02656$

$\sigma_1 = +4.60693$ and $\sigma_2 = -5.3125$

Nodal forces, unbalanced forces and residuals may be calculated to give the condensed variation factors as follows:

$$\begin{bmatrix} R^3_{c1} \\ R^3_{c2} \end{bmatrix} = \begin{bmatrix} -0.39307 \\ +0.08057 \end{bmatrix}$$

Again the nodal displacements may be calculated using equation (4.2b) giving:

$$\begin{bmatrix} \delta^3_1 \\ \delta^3_2 \\ \delta^3_3 \end{bmatrix} = \begin{bmatrix} \delta^2_1 \\ \delta^2_2 \\ \delta^2_3 \end{bmatrix} + \begin{bmatrix} \delta_{11} & \delta_{12} \\ \delta_{21} & \delta_{22} \\ \delta_{31} & \delta_{32} \end{bmatrix} \begin{bmatrix} R^3_{c1} \\ R^3_{c2} \end{bmatrix}$$

$$= \begin{bmatrix} 0 \\ 0.02656 \\ 0 \end{bmatrix} + \begin{bmatrix} 0 & 0 \\ 0 & 0.0025 \\ 0 & 0 \end{bmatrix} \begin{bmatrix} -0.39307 \\ +0.08057 \end{bmatrix}$$

$$= \begin{bmatrix} 0 \\ 0.02676 \\ 0 \end{bmatrix}$$

The technique converges to the correct answers and when compared with the usual procedure of the initial stiffness technique gives the same answers for each iteration. The full solution to nine decimal places is shown in table IV.1 at the back of this appendix.

IV.2.2 Tangential stiffness technique

The nodal displacements for the first iteration are $\{\delta^1\}=\{0,0.025,0\}$ obtained from the initial analysis. The element strains and stresses are calculated from the first iteration as before ($\varepsilon_1=+0.025$, $\varepsilon_2=-0.025$, $\sigma_1=+4.375$ and $\sigma_2=-5.0$). The new stiffness of element 1 is calculated using equations (IV.1) and (IV.5) to give:

$$[K^e] = \frac{AE_o}{L}(1 - 10\times0.025)\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \begin{bmatrix} 150 & -150 \\ -150 & 150 \end{bmatrix}$$

The stiffness of the second element will remain the same. The change in stiffness of each element may be expressed using equation (3.1):

$$\alpha^1 = \frac{150 - 200}{200} = -0.25 \quad \text{and} \quad \alpha^2 = \frac{200 - 200}{200} = 0$$

The changing variable considered for each element is E_T. However since the stiffness of only the first element is changing the analysis for unit loads at the nodes of the first element (shown in figures IV.1(iii) and IV.1(iv)) need be considered. The fractional changes at each end of the element may be expressed using equation (3.2), as follows:

$$\alpha_1 = \alpha_2 = \alpha^1 = -0.25$$

Unlike the initial stiffness technique, the unit load analyses must be updated to account for the change in stiffness of this element. This needs to be carried out for the unit loads shown in figure IV.1(iii) and (iv). Using equation (4.5a) and treating the unit load of figure IV.1(iv) as applied loading, gives the following equilibrium conditions:

$$\begin{bmatrix} 1+\alpha_1^2 f_{11} & \alpha_1^2 f_{12} \\ \alpha_2^2 f_{21} & 1+\alpha_2^2 f_{22} \end{bmatrix} \begin{bmatrix} R^2_{cI(12)} \\ R^2_{cI(22)} \end{bmatrix} + \begin{bmatrix} \alpha_1^2 f_{12} \\ \alpha_2^2 f_{22} \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \end{bmatrix}$$

Substituting and solving gives:

$$\begin{bmatrix} R^2_{cI(12)} \\ R^2_{cI(22)} \end{bmatrix} = \begin{bmatrix} -0.14286 \\ +0.14286 \end{bmatrix}$$

If the unit load of figure IV.1(iii) is similarly treated, then the condensed variation factors for this unit load are $\{R^2_{cI(11)}, R^2_{cI(21)}\}=\{0,0\}$. Hence the matrix of condensed variation factors for the unit loads, $[R^2_{cI}]$ is given by:

$$\begin{bmatrix} R^2_{cI(11)} & R^2_{cI(12)} \\ R^2_{cI(21)} & R^2_{cI(22)} \end{bmatrix} = \begin{bmatrix} 0 & -0.14286 \\ 0 & +0.14286 \end{bmatrix}$$

These condensed variation factors for the unit loads are now used to update the nodal displacements due to the unit loads given by equation (4.5b):

$$\begin{bmatrix} \delta^2_{11} & \delta^2_{12} \\ \delta^2_{21} & \delta^2_{22} \\ \delta^2_{31} & \delta^2_{32} \end{bmatrix} = \begin{bmatrix} \delta_{11} & \delta_{12} \\ \delta_{21} & \delta_{22} \\ \delta_{31} & \delta_{32} \end{bmatrix} + \begin{bmatrix} \delta_{11} & \delta_{12} \\ \delta_{21} & \delta_{22} \\ \delta_{31} & \delta_{32} \end{bmatrix} \begin{bmatrix} R^2_{cI(11)} & R^2_{cI(12)} \\ R^2_{cI(21)} & R^2_{cI(22)} \end{bmatrix}$$

$$= \begin{bmatrix} 0 & 0 \\ 0 & 0.0025 \\ 0 & 0 \end{bmatrix} + \begin{bmatrix} 0 & 0 \\ 0 & 0.0025 \\ 0 & 0 \end{bmatrix} \begin{bmatrix} 0 & -0.14286 \\ 0 & +0.14286 \end{bmatrix}$$

$$= \begin{bmatrix} 0 & 0 \\ 0 & 0.00286 \\ 0 & 0 \end{bmatrix}$$

The condensed variation factors for the applied loads are given by the residual forces for the second iteration of the initial stiffness technique as before:

$$\begin{bmatrix} R^2_{c1} \\ R^2_{c2} \end{bmatrix} = \begin{bmatrix} -0.625 \\ +0.625 \end{bmatrix}$$

The nodal displacements are calculated using equation (4.6b) with the updated nodal displacements due to the unit load analyses, to give:

$$\begin{bmatrix} \delta_1^2 \\ \delta_2^2 \\ \delta_3^2 \end{bmatrix} = \begin{bmatrix} \delta_1^1 \\ \delta_2^1 \\ \delta_3^1 \end{bmatrix} + \begin{bmatrix} \delta_{11}^2 & \delta_{12}^2 \\ \delta_{21}^2 & \delta_{22}^2 \\ \delta_{31}^2 & \delta_{32}^2 \end{bmatrix} \begin{bmatrix} R_{c1}^2 \\ R_{c2}^2 \end{bmatrix}$$

$$= \begin{bmatrix} 0 \\ 0.025 \\ 0 \end{bmatrix} + \begin{bmatrix} 0 & 0 \\ 0 & 0.00286 \\ 0 & 0 \end{bmatrix} \begin{bmatrix} -0.625 \\ +0.625 \end{bmatrix} = \begin{bmatrix} 0 \\ 0.02679 \\ 0 \end{bmatrix}$$

This completes the second iteration of the tangential stiffness technique. From this iteration the strains, stresses and changes in stiffness may be recalculated. The unit load analyses may be modified again using equations (4.5a) and (4.5b). The residual forces may be calculated and used as the condensed variation factors for the applied loads as above.

The results obtained using this technique are exactly the same as the usual procedure. Table IV.1 shows that the technique converges to the correct answers.

IV.3 Summary

The initial and tangential stiffness techniques were illustrated by reference to a simple one-dimensional finite element problem. It must be emphasized that these techniques are only applicable if the modified element stiffnesses are a scalar multiple of the original stiffness. If the stiffness of only a few elements is changing then the theorems of structural variation are obviously more efficient.

	Analysis of structure in figure IV.1 Displacement of node 2	
Iteration	Initial stiffness	Tangential stiffness
1	0.025000000	0.025000000
2	0.265625000	0.026785714
3	0.026763916	0.026794919
4	0.026790768	0.026794919
5	0.026794363	
6	0.026794845	
7	0.026794909	
8	0.026794918	
9	0.026794919	
10	0.026794919	

Table IV.1 Results of analyses using the theorems of structural variation

APPENDIX V

CONVERSION FACTORS

In some of the examples in this thesis, the units used were in the foot-pound system instead of the S.I. system. This was because the original examples obtained from the various papers cited used this particular system. The results obtained from the computer programs that were developed can then be easily compared with those in the papers with minimal difficulties and confusion in conversion. However the following table is provided to convert the foot-pound system to the S.I. system if this is required.

1 in $\equiv$ 25.4 mm $\equiv$ 2.54 cm $\equiv$ 0.0254 m

1 ft $\equiv$ 304.8 mm $\equiv$ 30.48 cm $\equiv$ 0.3048 m

1 in^2 $\equiv$ 645.16 mm^2 $\equiv$ 6.4516 cm^2 $\equiv$ 6.4516×10^{-4} m^2

1 ft^2 $\equiv$ 92903.04 mm^2 $\equiv$ 929.0304 cm^2 $\equiv$ 0.09290 m^2

1 in^3 $\equiv$ 16.39×10^3 mm^3 $\equiv$ 16.3871 cm^3 $\equiv$ 1.6387×10^{-5} m^3

1 ft^3 $\equiv$ 28.32×10^6 mm^3 $\equiv$ 28.32×10^3 cm^3 $\equiv$ 0.02832 m^3

1 lb $\equiv$ 0.4536 kg $\equiv$ 4.4482 N $\equiv$ 4.5359×10^{-4} t

1 p.s.i. or lb/in^2 $\equiv$ 6895 N/m^2 $\equiv$ 6.895×10^{-3} N/mm^2

1 p.s.f. or lb/ft^2 $\equiv$ 47.8803 N/m^2 $\equiv$ 4.788×10^{-5} N/mm^2

1 p.c.f. or lb/ft^3 $\equiv$ 157.0877 N/m^3 $\equiv$ 1.5709×10^{-7} N/mm^3

APPENDIX VI

COMPUTER PROGRAMS

VI.1 Introduction

A considerable proportion of the time spent on this research was spent on computer programming. The computer programs were developed for linear and nonlinear finite element analysis. For the linear analysis two sets of programs were prepared. The first set used the usual procedure of analysis. The second set were for reanalysis using the theorems of structural or geometric variation. The theory behind these two sets of programs may be found in section 2.2 and Chapter 3 respectively.

Similarly for the nonlinear analysis, two sets of programs were developed. The first set solves the nonlinear equilibrium equations by the Newton-Raphson method and its degenerate forms. The second set solves these equations using the theorems of geometric variation. The relevant theory for these programs are given in section 2.3 and Chapter 4 respectively.

By using two sets of programs, the CPU time may then be directly compared for the efficiency tests that were carried out in Chapters 5, 6, 7 and 8. The solutions obtained by the theorems of structural or geometric variation may also be compared with those from the usual procedures. This comparison will indicate whether any round-off errors or computational difficulties were encountered with the proposed techniques.

It is impossible to give a line by line account of the computer programs here. To facilitate understanding flowcharts are given to illustrate the algorithms. The flowcharts are rather general because too much detail might obscure the proposed solution algorithms. The objective is to show that the proposed techniques may easily be implemented with existing computer programs without any difficulty.

VI.2 Computer used, program writing and efficiency

The computer used was the general-purpose mainframe computer model ICL 2900 manufactured by International Computers Limited. The operating system is called EMAS 2900 which is a general-purpose and multi-access system developed by ERCC (Edinburgh Regional Computing Centre). Each user of the system is allocated with a processor and shares the computer with other users. The author's own processor consists of 10 megabytes of virtual memory which is sufficient to run the finite element programs. All the programs developed were coded in Fortran77 and double precision was used in the analysis. Although some of the programs were obtained elsewhere, in general the art of writing intelligible programs suggested by Meek et.al.[90] was used as a guide. The use of the finite element method and the theorems of structural or geometric variation involves a large amount of computation and many data transfers. Therefore it is essential that the programs must be written in an efficient form. The general rules of writing efficient programs given by Metcalf[96] and Muxworthy[99] are followed. ERCC has also provided a compiler which optimized the object code of the source program. This feature was used during the compilations, however details of this particular compiler are not available from ERCC.

The programs were all divided into subprograms linked together by a master program. The master program controls the execution of each subprogram. The use of such a programming technique is that it makes it easy to measure the CPU time to execute a particular segment of the program. This serves two purposes; the first one will indicate which parts of the programs are inefficient or poorly coded and secondly which part takes a large proportion of the total CPU time.

Obviously it is a waste of time to develop the programs from scratch. Many finite element programs are already available and those from Hinton and Owen[53,54,112] in particular form the basic building blocks of the programs used in this thesis. The programs from these authors are subsequently modified for the proposed solution techniques in linear and nonlinear finite element analysis. All the programs make minimal use of back-up storage except to store the element stiffness and [D][B] matrices. This will save a considerable amount of computing time.

VI.3 Linear finite element analysis

The formulations and derivations have already been given in section 2.2. The programs developed by Hinton and Owen[54] were used as the basis of the proposed techniques. These programs are well structured and divided into subprograms which are easy to understand. Therefore the modifications to these programs for the theorems of structural and geometric variation may be undertaken without completely rewriting the programs. The frontal solution technique used by Hinton and Owen[54] is replaced by the banded solution technique[53] written by the same authors. The banded solution technique is explained briefly in sections 2.2.2 and 2.2.3.

The program names for linear finite element analysis are tabulated in table VI.1. The program size is the total memory required to run the program. Most of the space is taken by the large arrays that are declared in these programs. Program listings are not included in this thesis because they would take too many pages to list.

Program name	Program size (Kbytes)	Element used	Type of analysis
TRIFINITE1	801.54	CST	Usual procedure
QUADFINITE1	808.14	QUAD4	Usual procedure
QUADFINITE2	842.15	QUAD8	Usual procedure

Table VI.1 Computer programs for linear analysis

The programs in table VI.1 are used in the comparative studies of the efficiency with the theorems of structural and geometric variation in Chapters 5 and 6 respectively.

VI.3.1 Programs using the theorems of structural variation

The programs in table VI.1 require modification for reanalysis using the theorems. In the computer implementation these are added to the existing programs as separate subprograms. These subprograms are described below:

1. Input data to identify which elements are to be modified and their new elastic modulus.

2. Set-up the matrix of unit and applied loads as given by equation (3.7)

3. Solve equation (3.7) for the displacements and reactions due to the multiple load cases. This is done simultaneously for efficiency.

4. Set-up the equilibrium equations involving the modified elements given by equation (3.27a). This is formed by adding the contribution of each modified element from equation (3.25).

5. Solve for the condensed variation factors by Gaussian elimination (without partial pivoting).

6. Evaluate the response of the modified structure using equations (3.28), (3.29), (3.13) and (3.15) for the displacements, reactions, strains and stresses respectively.

This is the most efficient version of the proposed technique by the direct evaluation of the condensed variation factors as proved in Chapter 5. Any type of element may be used in the technique and the general flowchart is illustrated in figure VI.1.

The flowchart provided is a straight forward application of the theorems. However further modifications may be undertaken by providing a 'loop backwards' at the end of the flowchart to reanalyse another modified structure. This is particularly useful in computer-aided-design where many alternative designs are needed for analysis. The complete list of programs are tabulated in table VI.2.

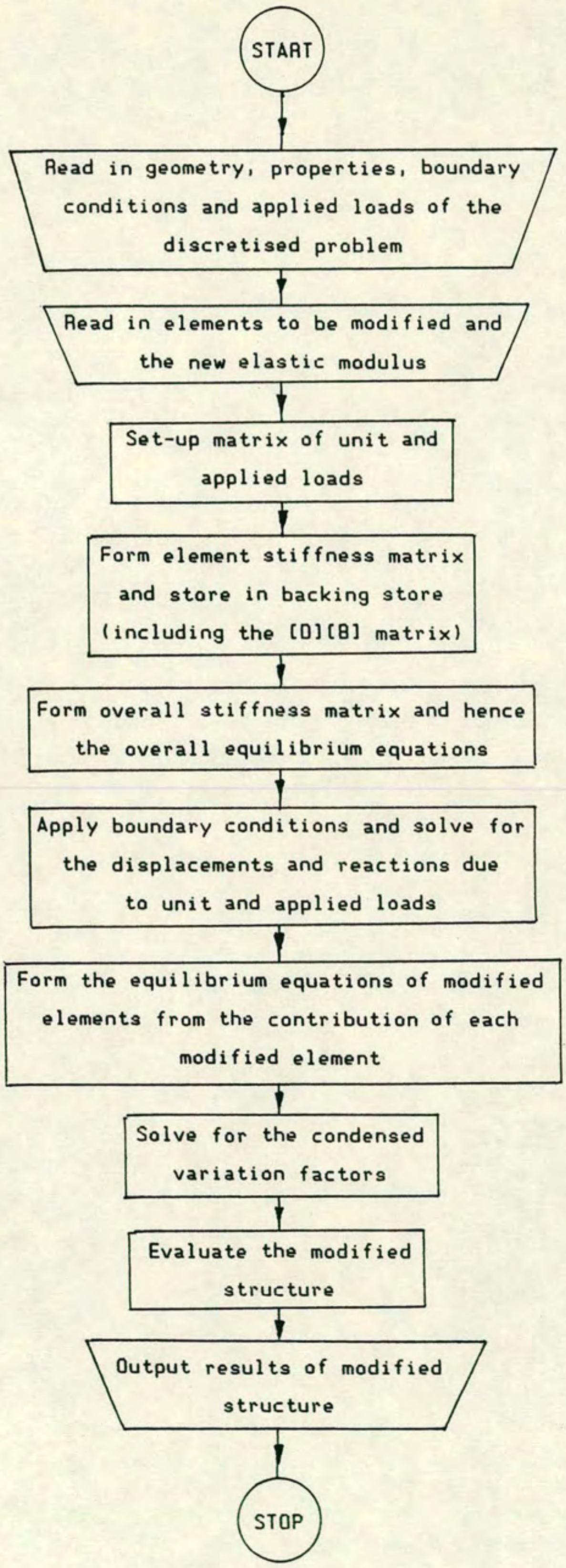


Figure VI.1 Flowchart for reanalysis by the theorems

Program name	Program size (Kbytes)	Element used for initial analysis	Element used for reanalysis
TRY1	1904.20	CST	CST(3.3)
TRY12	3185.23	CST	CST(3.10)
TRY2	1817.48	CST	NST(3.3)
TRY21	2040.14	CST	NST(3.10)
QUAD12	1811.00	QUAD4	QUAD4(3.10)
QUAD22	1840.64	QUAD8	QUAD8(3.10)

Table VI.2 Computer programs for reanalysis

For the shear wall analysis in section 5.3.1 the programs QUADFINITE1 and QUAD12 were used for comparison of the CPU times. In the soil excavation of section 5.3.2, the programs QUADFINITE1 and QUAD12 were slightly modified for evaluating the initial stresses and the incremental displacements and stresses. The names of these programs are tabulated in table VI.3.

Program name	Program size (Kbytes)	Element used in the analysis	Type of analysis
QUADGEO1	467.29	QUAD4	Usual procedure
QUADGEO2	1876.44	QUAD4	Using the theorems

Table VI.3 Computer programs for soil excavation

The programs developed for the theorems of structural variation require extra storage space. This may be observed from tables VI.1 and VI.2 where the program size gives a measure of the storage requirements. The extra storage space is required because of the multiple loading cases given by equation (3.7) which occupies a large proportion of the total memory. Some savings in computer storage was possible by using the array previously used for the structure stiffness matrix, for the equilibrium equations of the modified elements which must be solved for the variation factors. The use of a large program may not be a problem in computers with virtual memory but it may be very expensive.

VI.3.2 Programs using the theorems of geometric variation

The modifications are similar to that as discussed in section VI.3.1. The additional subprograms that are required are described below:

1. Input data to identify which nodes are to be **varied** and their new coordinates.

2. The number of affected elements, affected nodes 'a' and unaffected nodes 'u' may then be identified. The matrix of the unit loads at the 'a' and 'u' nodes given by equation (3.47) is set-up.

3. Solve equation (3.47) due to the multiple load cases and obtain the displacements and reactions due to the unit loads.

4. Form the equilibrium equations due to the affected elements given by equation (3.52a). This is formed by adding the contribution of each affected element from equation (3.50).

5. Solve for the scale factors by Gaussian elimination (without partial pivoting to save time).

6. Evaluate the response of the modified structure using equations (3.53), (3.54), (3.55) and (3.56) for the displacements, reactions, strains and stresses respectively.

The general flowchart of the technique is illustrated in figure VI.2. The programs that were developed are tabulated in table

VI.4. These programs are used in the comparative tests of sections 6.2 and 6.3.

Program name	Program size (Kbytes)	Element used for initial analysis	Element used for reanalysis
TRYGEOMTR1	1244.12	CST	NST(3.32)
TRYGEOMTR2	1240.69	CST	CST & NST(3.32)†
TRYGEOMTR3	1209.82	CST	CST(3.50)
QUADGEOMTR1	1234.50	QUAD4	QUAD4(3.50)
QUADGEOMTR2	1284.41	QUAD8	QUAD8(3.50)

Table VI.4 Computer programs for reanalysis

† Note that in equation (3.32) the terms $f_{i,jh}\sin\theta_{ik}$ and $f_{i,jh}\cos\theta_{ik}$ are replaced by the nodal forces in the x and y directions respectively.

For the shape optimization of section 6.4.1 the programs QUADFINITE2 and QUADGEOMTR2 were used for comparison of the CPU times taken. In addition these programs have a segment which extrapolates the stresses evaluated at the Gauss points to the nodes of the elements. This technique of obtaining the stresses at the nodes is suggested by Hinton and Campbell[52]. For the seepage analysis in section 6.4.2 two separate programs were developed. The analysis is now for one degree of freedom and the permeability matrix is used instead of the elasticity matrix. Otherwise the same set of routines developed previously may be used with minor modifications. The flowchart for the version of the program by the theorems of geometric variation is given in figure VI.3. The program names and sizes are given in table VI.5.

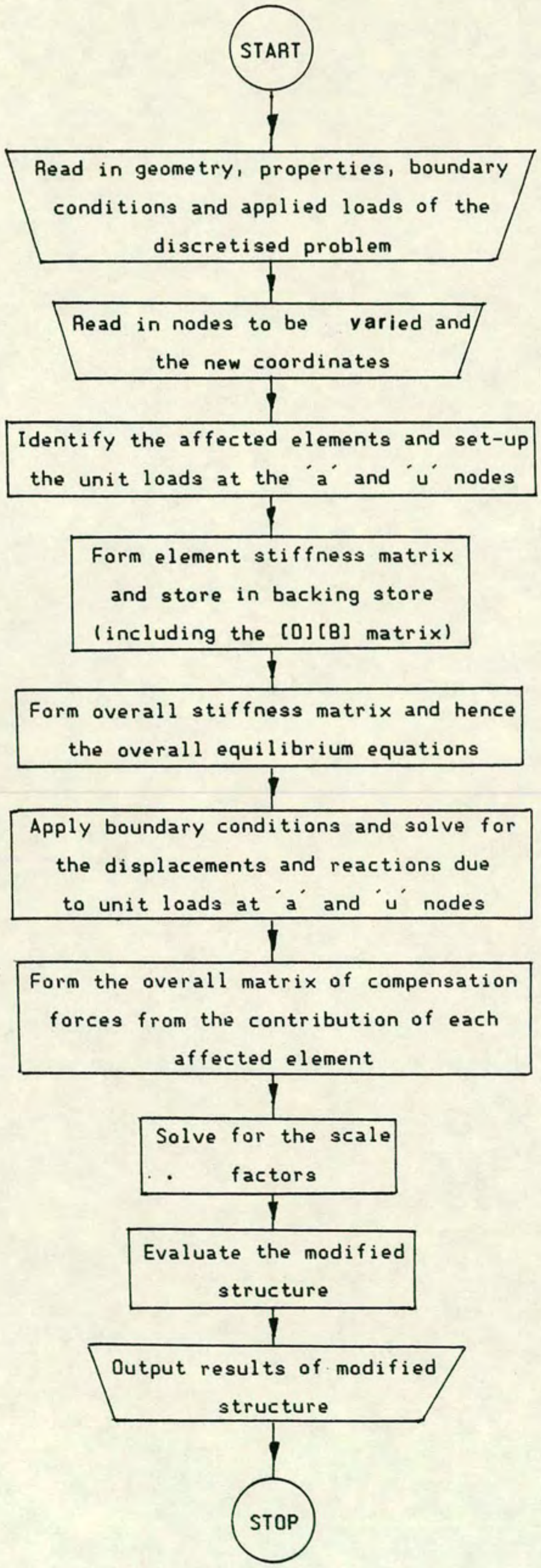


Figure VI.2 Flowchart for reanalysis by the geometric theorems

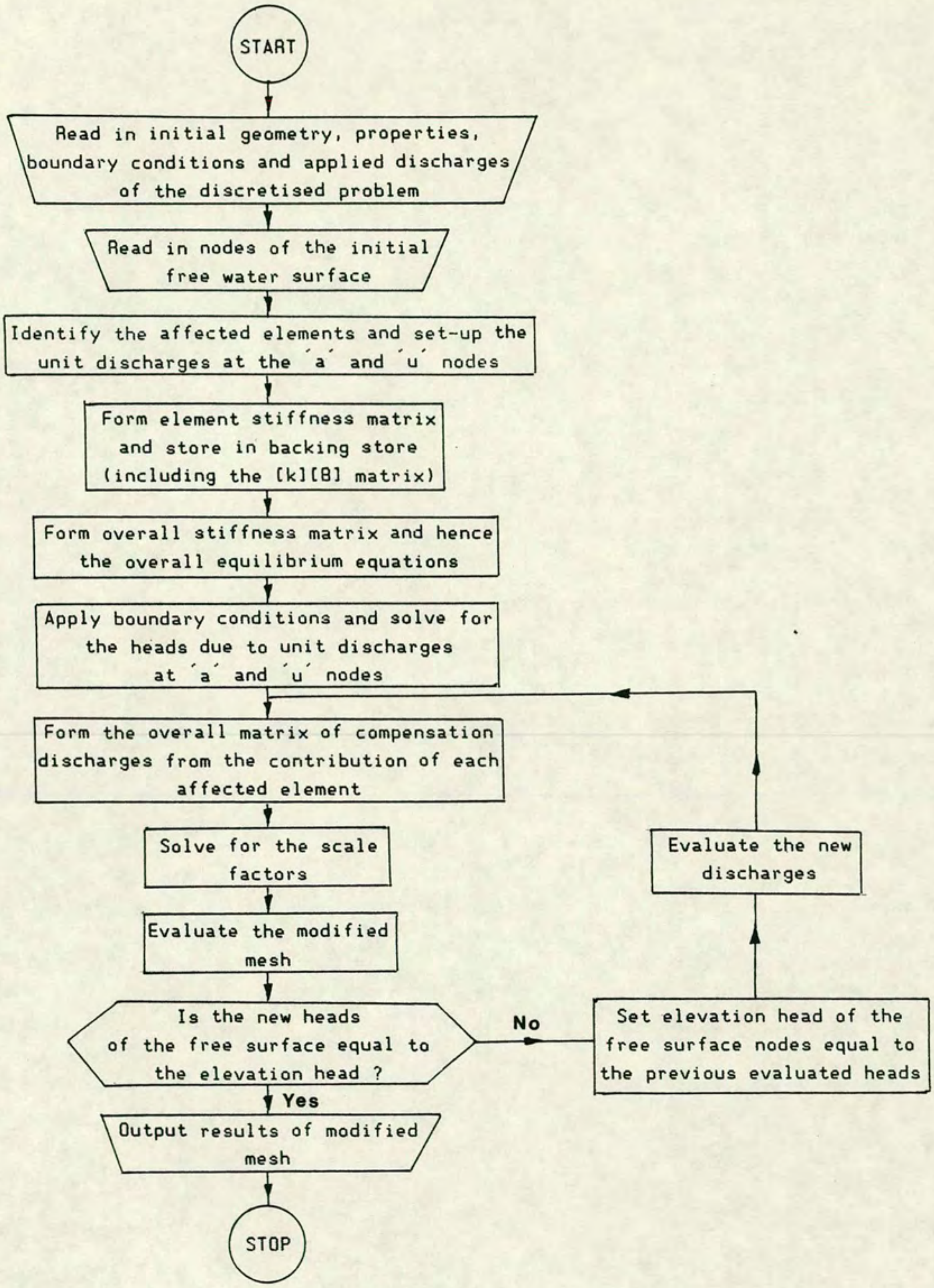


Figure VI.3 Flowchart for seepage analysis

Program name	Program size (Kbytes)	Element used in the analysis	Type of analysis
QUADFLOW1	824.94	QUAD4	Usual procedure
QGEOFLOW1	1310.13	QUAD4	Geometric theorems

Table VI.5 Computer programs for seepage analysis

The theorems of geometric variation in general require extra storage space as may be seen by comparing table VI.1 with VI.4.

VI.4 Nonlinear finite element analysis

The computer implementation of material nonlinear analysis was obtained from Owen and Hinton[112]. This program was modified by replacing the frontal solver with the band solver[53]. The Newton-Raphson methods used for solving the nonlinear equilibrium equations are the IS, TS, TSV, ITS, ITSV and ITS2 techniques. These techniques have been described in section 2.4 as well as in Chapters 7 and 8.

Owen and Hinton[112] also provided a program for dynamic geometrical nonlinear analysis. This program was modified for static analysis using the same band solver of the material nonlinear analysis program. The nonlinear solution techniques of Newton-Raphson are used in the algorithm.

For combined material/geometrical nonlinear problems, the extension of the above programs is fairly simple. The same set of subprograms are used from the two programs to form the material/geometrical nonlinear program. This is valid so long as the strains are assumed small. Table VI.6 lists the programs that were used for comparison with the theorems of geometric variation.

Program name	Program size (Kbytes)	Type of nonlinear problem	Newton-Raphson method used
PLAST1X	100.63	Material	IS technique
PLAST2X	96.26		TS technique
PLAST2Y	96.24		TSV technique
PLAST3X	101.27		ITS technique
PLAST3Y	101.53		ITSV technique
PLAST4X	101.31		ITS2 technique
GEOMT1X	99.32	Geometrical	IS technique
GEOMT2X	93.91		TS technique
GEOMT2Y	93.95		TSV technique
GEOMT3X	99.19		ITS technique
GEOMT3Y	99.65		ITSV technique
GEOMT4X	99.23		ITS2 technique
PLASGEO1X	116.08	Material/ geometrical	IS technique
PLASGEO2X	111.49		TS technique
PLASGEO2Y	111.35		TSV technique
PLASGEO3X	116.72		ITS technique
PLASGEO3Y	117.22		ITSV technique
PLASGEO4X	116.77		ITS2 technique

Table VI.6 Computer programs for nonlinear analysis

VI.4.1 Programs for material nonlinear analysis by the theorems of geometric variation

The programs of Owen and Hinton[112] that are tabulated in table VI.6 may easily be modified for the theorems of geometric variation. This was undertaken by additional subprograms that are described below:

1. Input data for identification of which elements likely to

be plastic.

2. Identification of the affected nodes, 'a' and unaffected nodes, 'u'. The total number of unit loads required may then be obtained and hence set-up the loading matrix.

3. Solution of the structure stiffness equations with multiple loading cases as given by equation (4.10).

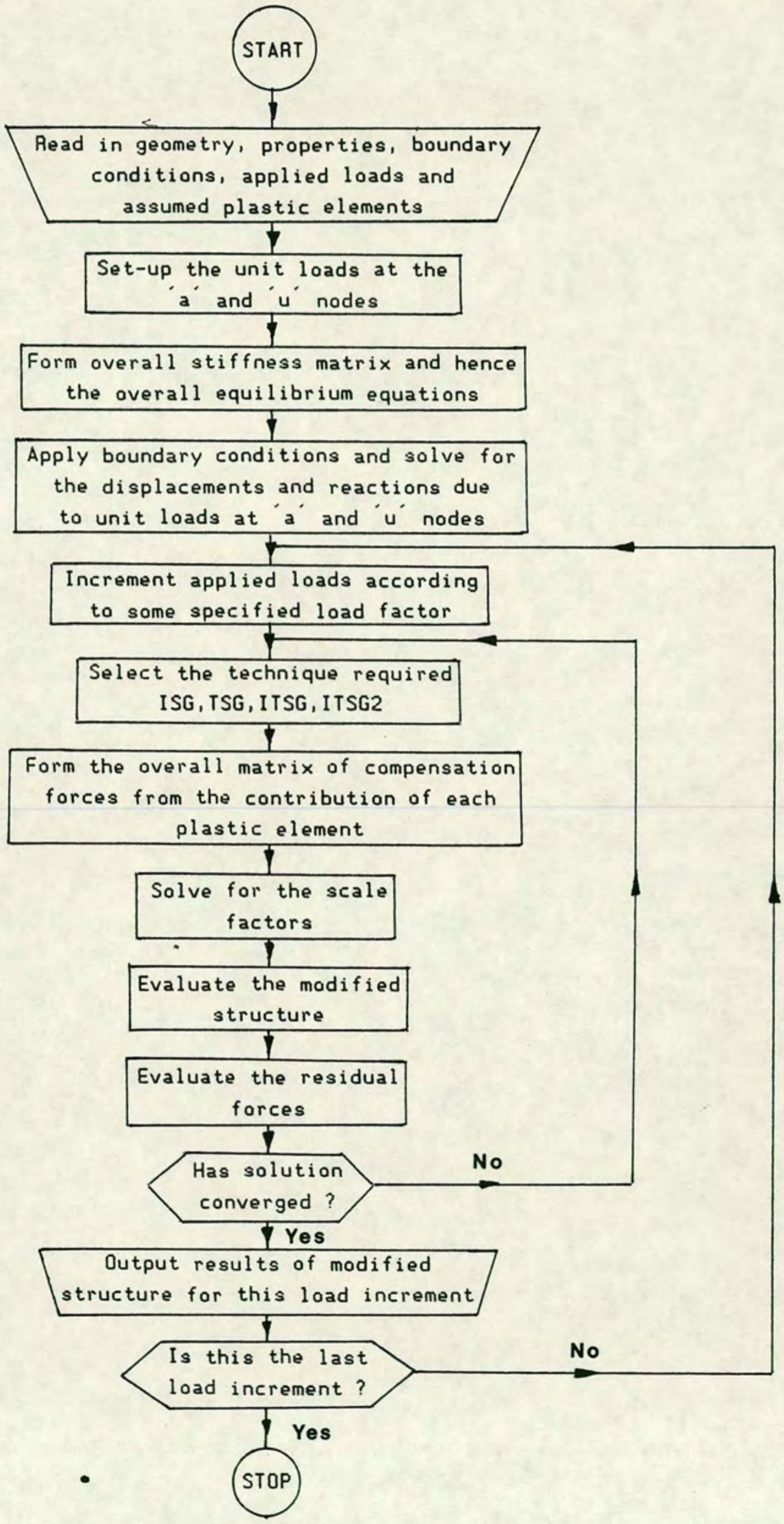
4. Set-up matrix of overall compensation forces as in equation (4.12c) or (4.15c) depending on the nonlinear solution technique selected. Equation (4.15c) is formed by adding in the contribution of each plastic element from equation (4.13). The plastic elements are identified by the presence of plastic strains at the Gauss points. This subprogram is not required if the initial stiffness technique is used.

5. Solve for the scale factors using Gaussian elimination for a full unsymmetric matrix. No partial pivoting was undertaken here so that the program is efficient. This is not required by the initial stiffness technique.

6. Evaluate the response of the modified structure using equation (4.12a) for the displacements.

In nonlinear analysis at the end of step 6 the residual forces are evaluated from equation (4.12b). If this not zero or within a specified tolerance the residual forces are treated as applied loads. Steps 4 to 6 are then repeated. For the variant forms of the technique the unit loads are those at the 'a' nodes and the total applied loads as discussed in section 4.2.2.4.

The flowchart for the nonlinear analysis using the theorems of geometric variation is in figure VI.4. It is an adaptation of the flowchart of Owen and Hinton[112]. The programs that were developed for comparison, with the Newton-Raphson methods in table VI.6 are tabulated in table VI.7.



Note that for the variant forms the flowchart is similar

Figure VI.4 Flowchart for material nonlinear analysis

Program name	Program size (Kbytes)	Theorems of geometric variation
PLAST11X	1321.88	ISG technique
PLAST11Y	1320.98	ISVG technique
PLAST22X	1394.64	TSG technique
PLAST22Y	1391.15	TSVG technique
PLAST33X	1441.54	ITSG technique
PLAST33Y	1440.12	ITSVG technique
PLAST44X	1441.79	ITSG2 technique

Table VI.7 Computer programs for material nonlinear analysis

The programs in table VI.7 are used for the comparative tests in Chapter 7.

VI.4.2 Programs for geometrical nonlinear analysis by the theorems of geometric variation

A similar set of subprograms to that of those in section VI.4.1 are needed for geometrical nonlinear analysis. These are outlined below:

1. Input data for assumed nonlinear elements. In addition the factor d_f is specified depending on the type of analysis required as discussed in Chapter 8.

2. The same as step 2 in section VI.4.1

3. The same as step 3 in section VI.4.1

4. The same as step 4 in section VI.4.1. The nonlinear elements are identified when the their nodal displacements exceed the value of the specified d_f.

5. The same as step 5 in section VI.4.1.

6. The same as step 6 in section VI.4.1.

Steps 4 to 6 are repeated if the residual forces are not zero or within some specified tolerance. If the variant techniques are used unit load analyses are required at the 'a' nodes and the total applied loads as discussed in section 4.2.2.4.

A general flowchart of the solution algorithm is in figure VI.5. The programs that were developed for comparison with the Newton-Raphson methods in table VI.6 are tabulated in table VI.8.

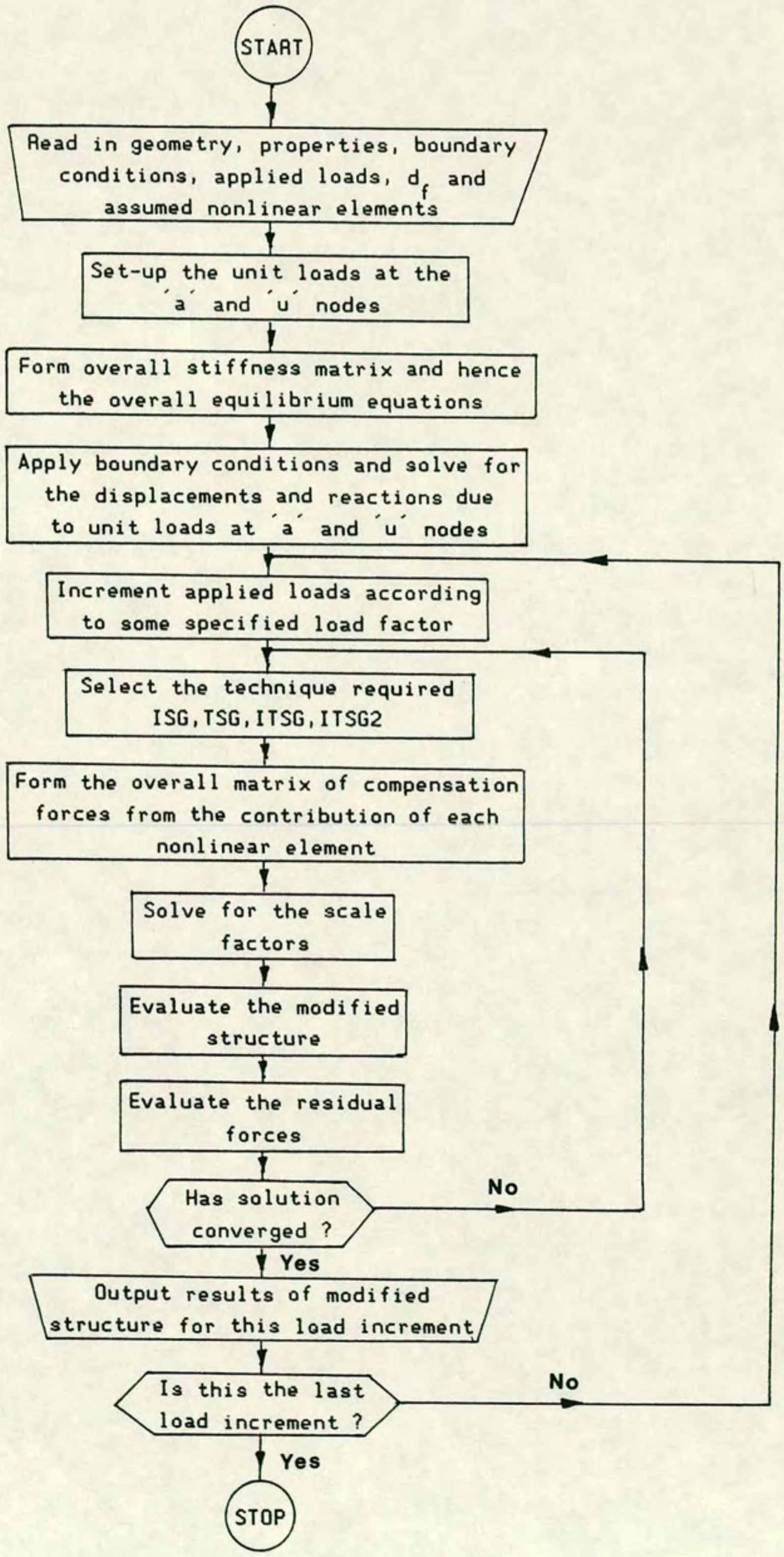
Program name	Program size (Kbytes)	Theorems of geometric variation
GEOMT11X	1339.58	ISG technique
GEOMT11Y	1338.54	ISVG technique
GEOMT22X	1492.63	TSG technique
GEOMT22Y	1488.89	TSVG technique
GEOMT33X	1477.93	ITSG technique
GEOMT33Y	1476.32	ITSVG technique
GEOMT44X	1478.23	ITSG2 technique

Table VI.8 Computer programs for geometrical nonlinear analysis

The programs in table VI.8 are used for the comparative tests in Chapter 8.

VI.4.3 Programs for material/geometrical nonlinear analysis by the theorems of geometric variation

These programs are an extension of the programs in section VI.4.2. Instead of using the elasticity matrix, the elasto-plastic matrix is used when the yield stress of the material is exceeded. The subprograms for the elasto-plastic matrix are obtained from section VI.4.1. The flowcharts and the modifications required are similar to that in section VI.4.2. Note that the programs are valid for such problems



Note that for the variant forms the flowchart is similar

Figure VI.5 Flowchart for geometrical nonlinear analysis provided that the strains are assumed small. The programs are tabulated in table VI.9.

Program name	Program size (Kbytes)	Theorems of geometric variation
PLASGEO11X	1356.82	ISG technique
PLASGEO11Y	1355.79	ISVG technique
PLASGEO22X	1511.57	TSG technique
PLASGEO22Y	1507.82	TSVG technique
PLASGEO33X	1496.88	ITSG technique
PLASGEO33Y	1495.41	ITSVG technique
PLASGEO44X	1497.17	ITSG2 technique

Table VI.9 Computer programs for mater./geomet. nonlinear analysis

The programs in table VI.9 are used in Chapter 9.

VI.5 Pre- and post-processing programs

In finite element analysis the use of pre- and post-processing programs are essential aids for understanding the structural behaviour. The pre-processing program is mainly concern with the generation of the input data. This will save time in data preparation by hand. The post-processing program is used to plot the deformed structure, stress contours and principal stresses. Most finite element output are voluminous and to comprehend pages of computer printout is a mighty task. Plots of these output will quickly give an idea of the critical design requirements at a glance.

In table VI.10 a list of pre- and post-processing programs used in this research with a brief description are given.

Program name	Program description
PRODMESH1	Generates finite element mesh for CST, QUAD4 and QUAD8 elements. A plot of the generated mesh is provided.
PRODRAW1	Reads in finite element data and plots the finite element mesh. Useful for checking data if it is generated by hand.
PRODRAW2	Renumber the nodes of the finite element mesh and plots the mesh.
PRODRAW3	For plotting of the principal stresses.
PRODRAW4	For plotting of the deformed finite element mesh.
GPCP	Software for plotting the stress contours. The source program is not available, see manual[23].

Table VI.10 Computer programs for pre- and post processing

VI.5.1 Programs for mesh generation and bandwidth reduction

The PRODMESH1 program is used to generate the finite element mesh for different type of elements. It also has the ability to grade the mesh generated, when a finer mesh is needed near regions of high stress gradients. Away from the region of interest a coarser mesh will usually be sufficient. This program for generating the CST, QUAD4 and QUAD8 elements was obtained from Hinton and Owen[53]. The finite element mesh generated by using this program was extensively used in this thesis. The program was extended for plotting the finite element mesh with the elements and nodes numbered using the graphics software[36] provided by ERCC.

Usually the mesh generated will not result in an optimum node numbering system. The semi-bandwidth in such cases will be large and hence the reduction of the stiffness matrix is inefficient. The program PRODRAW2 developed by Collins[32] was used to renumber the nodes such that a smaller semi-bandwidth is achieved. However the semi-bandwidth obtained by this program is not guaranteed to be the optimum one.

VI.5.2 Programs for plotting of displacements and stresses

The post-processing programs PRODRAW3 and PRODRAW4 were used to plot the stresses and deformed mesh respectively. This is important from a design point of view as the results from a finite element analysis may quickly be assimilated. Regions of high stress gradients and excessive deflections may also be easily identified.

The GPCP program in table VI.10 was used to plot the stress contours. However sample plots are not given in this thesis as it is irrelevant.

VI.6 Final comments

The programs developed used the basic subprograms of Hinton and Owen[53,54,112]. The documentation of these subprograms are explained in their texts and it is pointless to duplicate their efforts. The main aim was to show that the theorems of structural and geometric variation may easily be incorporated into existing computer codes. The task is very simple if the existing codes are highly structured and divided into modules. These theorems may then be developed separately and 'plugged-in' with minimal impact at other program levels.

Although the programs developed here were used in analysis, their primary aim was for studying the feasibility of the theorems of structural and geometric variation. They are not production-level programs to be used for processing large scale problems. This is impossible to achieve here because it requires many man-hours of programming effort. However it should be appreciated that the proposed techniques may easily be implemented into existing codes. This was clearly demonstrated by the modification of the computer codes of Hinton and Owen[53,54,112].

REFERENCES

1. ARGYRIS,J.H., "On the analysis of complex elastic structures", Applied Mechanics Reviews, Vol.11, No.7, pp 331-338, July 1958

2. ARGYRIS,J.H., "Recent Advances in Matrix Methods of Structural Analysis in Aeronautical Science", Vol.4, Pergamon Press, London, 1963

3. ARGYRIS,J.H., "The matrix analysis of structures with cut outs and modifications", 9 th. Int. Cong. of Applied Mechanics, Univ. of Brussels, Belgium, Vol.6, pp 131-140, Sept. 1956

4. ARGYRIS,J.H.,BRONLUND,O.E.,ROY,J.R.,SCHARPF,D.W., "A direct modification procedure for the displacement method", AIAA Journal, Vol.9, No.9, pp 1861-1864, Sept. 1971

5. ARGYRIS,J.H.,KELSEY,S., "Energy Theorems and Structural Analysis", Butterworths, London, 1960. (Reprinted from Aircraft Engineering, Vols.26-27, Oct. 1954 - May 1955)

6. ARGYRIS,J.H.,KELSEY,S., "Initial strains in the matrix force method of structural analysis", J. of the Royal Aero. Soc., Vol.64, pp 493-495, Aug. 1960

7. ARGYRIS,J.H.,KELSEY,S., "The matrix force method of structural analysis and some new applications", Great Britain Aeronautical Research Council Technical Report, R & M No.3034, Vol.93, pp 787-828, Feb. 1956

8. ARGYRIS,J.H.,ROY,J.R., "General treatment of structural modifications", J. of the Struct. Div., Proc. of the A.S.C.E., Vol.98, No.ST2, pp 465-492, Feb. 1972

9. ARMEN,H., "Applications of a substructuring technique to the problem of crack extension and closure", Report No.NASA-CR-132458, RE-480, Grumman Aerospace Corp., Bethpage, N.Y. Research Dept., July 1974

10. ARORA,J.S., "Survey of structural reanalysis techniques",

J. of the Struct. Div., Proc. of the A.S.C.E., Vol.102, No.ST4, pp 783-802, April 1976

11. ARORA,J.S.,RIM,K., "An algorithm for fail safe structural optimization and a review of reanalysis techniques", Tech. Rept. No.11, Dept. of Mechanics and Hydraulics, College of Engineering, Univ. of Iowa, March 1974

12. ATREK,E., "Theorems of structural variation : A simplification", Int. J. for Numerical Methods in Engr., Vol.21, pp 481-485, 1985

13. ATREK,E.,GALLAGHER,R.H.,RAGSDELL,K.M.,ZIENKIEWICZ,O.C., Eds., "New Directions in Optimum Structural Design", John Wiley & Sons, New York, 1984

14. BAKRY,M.A.E., "Optimal Design of Transmission Towers", A thesis submitted for the Degree of Doctor of Philosophy, Univ. of Surrey, England, 1978

15. BATHE,K.J.,KHOSHGOFTAAR,M.R., "Finite element free surface seepage analysis without mesh iteration", Int. J. Anal. & Num. Meth. in Geomechanics, Vol.3, pp 13-22, 1979

16. BATHE,K.J.,RAMM,E.,WILSON,E.L., "Finite element formulation for large deformation dynamic analysis", Int. J. for Numerical Methods in Engr., Vol.9, pp 353-386, 1975

17. BENNETT,J.M., "Triangular factors of modified matrices", Numerishe Mathematik, Vol.7, pp 217-221, 1965

18. BEST,G., "A method of structural weight minimization suitable for high-speed digital computers", AIAA Journal, Vol.1, No.2, pp 478-479, Feb. 1963

19. BEST,G.C., "The stress-area method applied to frames", J. of the Aero. Sci., Vol.13, No.3, pp 151-155, March 1946

20. BEST,G.C., "The stress-area method of designing beams", J. of the Aero. Sci., Vol.12, No.3, pp 298-304, July 1945

21. BHATIA,K.G., "Rapid iterative reanalysis for automated design", NASA TN D-7357, NASA, Washington D.C., Oct. 1973

22. BROCK,J.E., "Variation of coefficients of simultaneous linear equations", Quart. of Applied Maths., Vol.11, pp 234-240, 1953

23. CALCOMP, "GPCP - A General Purpose Contouring Program", California Computer Products Inc., User's Manual, August 1973

24. CELIK,T., "Nonlinear moment curvature analysis by means of theorems of structural variation", Istanbul Devlet Muhendislik ve Mimarlik Akademisi Dergisi, No.6, pp 81-103, 1981

25. CELIK,T., "The theorems of structural variation and their application in the elastic-plastic analysis of frames", Istanbul Devlet Muhendislik ve Mimarlik Akademisi Dergisi, No.5, pp 125-138, 1979

26. CHAN,H.F., "The Geometric Theorems of Structural Variation", 4th. year project submitted for the Degree of B.Sc., Edinburgh Univ., Scotland, 1983

27. CHANDRASEKARAN,V.S.,KING,G.J.W., "Simulation of excavation using finite elements", J. of the Geotech. Div., Proc. of the A.S.C.E., Vol.100, No.GT9, pp 1086-1089, Sept. 1974

28. CHEN,W.F.,SALEEB,A.F., "Constitutive Equations for Engineering Materials, Vol.1: Elasticity and Modeling", John Wiley & Sons, 1982

29. CICALA,P., "Effects of cutouts in semimonocoque structures", J. of the Aero. Sci., Vol.15, No.3, pp 171-179, March 1948

30. CLOUGH,R.W., "The finite element method after twenty-five years: A personal view", Computers & Structures, Vol.12, pp 361-370, 1980

31. CLOUGH,R.W., "The finite element method in plane stress analysis", A.S.C.E., J. Struct. Div., Proc. 2nd. Conf. Electronic Computation, pp 345-378, 1960

32. COLLINS,R.J., "Bandwidth reduction by automatic renumbering", Int. J. for Numerical Methods in Engr., Vol.6, pp 345-356, 1973

33. DAS,P.C., "Reanalysing structures with small modifications", Computer Aided Design, Vol.10, No.6, pp 371-374, Nov. 1978

34. DING,H.,GALLAGHER,R.H., "Approximate force method reanalysis techniques in structural optimization", Int. J. for Numerical Methods in Engr., Vol.21, pp 1253-1267, 1985

35. DUNCAN,J.M.,CHANG,C.Y., "Nonlinear analysis of stress and strain in soils", J. of the Soil Mech. & Found. Div., Proc. of the A.S.C.E., Vol.96, No.SM5, pp 1629-1653, Sept. 1970

36. EDINBURGH REGIONAL COMPUTING CENTRE, "ERCC Graphics Manual", 2nd. Edn., Sept. 1980

37. ERTAS,R.,FENVES,S.J., "Automatic analyser for iterative design", Civ. Eng. Studies, Structural Research Series n.352, Univ. of Illinois, Illinois, Sept. 1969

38. FENVES,S.J.,ERTAS,R., Discussion of "Multiple configuration analysis of structures", J. of the Struct. Div., Proc. of the A.S.C.E., Vol.95, No.ST7, pp 1586-1589, July 1969

39. FLETCHER,R.,POWELL,M.J.D., "On the modification of LDL^T factorizations", Mathematics of Computation, Vol.28, pp 1067-1087, 1974

40. FOX,R.L., "Optimization Methods for Engineering Design", Addison-Wesley Publishing Co., Reading, Massachusetts, 1971

41. FOX,R.L.,MIURA,H., "An approximate analysis technique for design calculations", AIAA Journal, Vol.9, No.1, pp 177-179, Jan. 1971

42. FRANCAVILLA,A.,RAMAKRISHNAN,C.V.,ZIENKIEWICZ,O.C., "Optimization of shape to minimize stress concentration", J. of Strain Analysis, Vol.10, No.2, pp 63-70, 1975

43. FRANCE,P.W.,PAREKH,C.J.,PETERS,J.C.,TAYLOR,C., "Numerical analysis of free surface seepage problems", J. of the Irr. & Drain. Div., Proc. of the A.S.C.E., Vol.57, No.IR1, pp 165-179, March 1971

44. GALLAGHER,R.H., "Computerized structural analysis and design - The next twenty years", Computers & Structures, Vol.7, pp 495-501, 1977

45. GILL,P.E.,GOLUB,G.H.,MURRAY,W.,SAUNDERS,M.A., "Methods for modifying matrix factorizations", Mathematics of Computation, Vol.28, pp 505-535, 1974

46. GILL,P.E.,MURRAY,W.,SAUNDERS,M.A., "Methods for computing and modifying the LDV factors of a matrix", Mathematics of Computation, Vol.29, pp 1051-1077, 1975

47. GIRIJAVALLABHAN,C.V., "Analysis of shear walls with openings", J. of the Struct. Div., Proc. of the A.S.C.E., Vol.95, No.ST10, pp 2093-2103, Oct. 1969

48. GOODEY,W.J., "Notes on a general method of treatment of structural discontinuities", J. of the Royal Aero. Soc., Vol.59, pp 695-697, Oct. 1955

49. GOODEY,W.J., "Solution of modified linear simultaneous equations", Aircraft Engineering, Vol.31, pp 358-359, 364, Dec. 1959

50. GRZEDZIELSKI,A.L.M., "Note on some applications of the matrix force method of structural analysis", J. of the Royal Aero. Soc., Vol.64, pp 354-357, June 1960

51. GRZEDZIELSKI,A.L.M.,ARGYRIS,J.H.,KELSEY,S., Discussion and comments on "The initial strain concept", J. of the Royal Aero. Soc., Vol.65, pp 127-138, Feb. 1961

52. HINTON,E.,CAMPBELL,J.S., "Local and global smoothing of discontinous finite element functions using a least squares method", Int. J. for Numerical Methods in Engr., Vol.8, pp 461-480, 1974

53. HINTON,E.,OWEN,D.R.J., "An Introduction to Finite Element Computations", Pineridge Press, Swansea, U.K., 1979

54. HINTON,E.,OWEN,D.R.J., "Finite Element Programming", Academic Press Inc., London, 1977

55. HIRAI,I.,WANG,B.P.,PILKEY,W.D., "An efficient zooming method for finite element analysis", Int. J. for Numerical Methods in Engr., Vol.20, pp 1671-1683, 1984

56. HOERNER,S.V., "Homologous deformations of tiltable telescopes", J. of the Struct. Div., Proc. of the A.S.C.E., Vol.93, No.ST5, pp 461-485, Oct. 1967

57. HOUSEHOLDER,A.S., "A survey of some closed methods for inverting matrices", SIAM Journal, Vol.5, No.3, pp 155-169, Sept. 1957

58. HOUSEHOLDER,A.S., "Principles of Numerical Analysis", McGraw-Hill Book Company, New York, 1953

59. HOUSEHOLDER,A.S., "The Theory of Matrices in Numerical Analysis", Blaisdell Pub. Co., New York, 1964

60. HUEBNER,K.H.,THORNTON,E.A., "The Finite Element Method for Engineers", 2nd. edn., John Wiley & Sons, New York, 1982

61. KANCHI,M.B.,ZIENKIEWICZ,O.C.,OWEN,D.R.J., "The visco-plastic approach to problems of plasticity and creep involving geometric non-linear effects", Int. J. for Numerical Methods in Engr., Vol.12, pp 169-181, 1978

62. KAVANAGH,K.T., "An approximate algorithm for the reanalysis of structures by the finite element method", Computers & Structures, Vol.2, pp 713-722, 1972

63. KAVLIE,D.,POWELL,G.H., "Efficient reanalysis of modified structures", J. of the Struct. Div., Proc. of the A.S.C.E., Vol.97, No.ST1, pp 377-392, Jan. 1971

64. KIRSCH,U., "Approximate structural reanalysis based on series expansion", Computer Methods in Applied Mechanics & Engr., Vol.26, pp 205-223, 1981

65. KIRSCH,U., "Approximate structural reanalysis for optimization along a line", Int. J. for Numerical Methods in Engr., Vol.18, pp 635-651, 1982

66. KIRSCH,U., "On some simplified models for optimal design of structural systems", Computer Methods in Applied Mechanics & Engr., Vol.48, pp 155-169, 1985

67. KIRSCH,U., "Optimal design based on approximate scaling", J. of the Struct. Div., Proc. of the A.S.C.E., Vol.108, No.ST4, pp 888-909, April 1982

68. KIRSCH,U., "Optimum Structural Design", McGraw-Hill Book Co., New York, 1981

69. KIRSCH,U.,HOFMAN,B., "Approximate behavior models for optimum structural design", Proc. Int. Symp. on Optimum Structural Design, Tuscon, Arizona, pp 7.17-7.26, 1981

70. KIRSCH,U.,RUBINSTEIN,M.F., "Modification of structural analysis by the solution of a reduced set of equations", UCLA Paper ENG-0570, Univ. of California, Los Angeles, Calif., Dec. 1970

71. KIRSCH,U.,RUBINSTEIN,M.F., "Reanalysis for limited structural design modifications", J. of the Engr. Mech. Div., Proc. of the A.S.C.E., Vol.98, No.EM1, pp 61-70, Feb. 1972

72. KIRSCH,U.,RUBINSTEIN,M.F., "Structural reanalysis by iteration", Computers & Structures, Vol.2, pp 497-510, 1972

73. KIRSCH,U.,TOLEDANO,G., "Approximate reanalysis for

modifications of structural geometry", Computers & Structures, Vol.16, pp 269-277, 1983

74. KLEIBER,M.,LUTOBORSKI,A., "Modified triangular factors in the incremental finite element analysis with nonsymmetric stiffness changes", Computers & Structures, Vol.9, pp 599-602, 1978

75. KOSKO,E., "Effect of local modifications in redundant structures", J. of the Aero. Sci., Vol.21, No.3, pp 206-207, March 1954

76. KOSKO,E., "Matrix inversion by partitioning", Aeronautical Quarterly, Vol.8, pp 157-184, May 1957

77. KOVACH,L.D., "Life can be so nonlinear", American Scientist, Vol.48, pp 218-225, 1960

78. LAM,H.L.,CHOI,K.K.,HAUG,E.J., "A sparse matrix finite element technique for iterative structural optimization", Computers & Structures, Vol.16, pp 289-295, 1983

79. LAW,K.H., "Sparse matrix factor modification in structural reanalysis", Int. J. for Numerical Methods in Engr., Vol.21, pp 37-63, 1985

80. LAW,K.H.,FENVES,S.J., "Sparse matrices, graph theory, and reanalysis", Proc. of the Int. Conf. in Civ. Eng., 1st., New York, N.Y., by ASCE, pp 234-249, May 12-14, 1981

81. MACNEAL,R.H., "Application of the compensation theorem to the modification of redundant structures", J. of the Aero. Sci., Vol.20, No.10, pp 726-727, Oct. 1953

82. MAJID,K.I., "Optimum Design of Structures", Newnes Butterworth, 1974

83. MAJID,K.I.,CELIK,T., "The elastic-plastic analysis of frames by the theorems of structural variation", Int. J. for Numerical Methods in Engr., Vol.21, pp 671-681, 1985

84. MAJID,K.I.,ELLIOTT,D.W.C., "Forces and deflections in changing structures", The Structural Engineer, Vol.51, No.3, pp 93-101, 1973

85. MAJID,K.I.,ELLIOTT,D.W.C., "Topological design of pin jointed structures by non-linear programming", Proc. of the Instn. of Civ. Eng., Vol.55, Part 2, pp 129-149, 1973

86. MAJID,K.I.,SAKA,M.P., "Optimum shape design of rigidly jointed frames", Proc. of the Symp. on the Appl. of Comp. Meth. in Engr., Vol.1, pp 521-531, Univ. of Southern California, 1977

87. MAJID,K.I.,SAKA,M.P.,CELIK,T., "The theorems of structural variation generalised for rigidly jointed frames", Proc. of the Instn. of Civ. Eng., Vol.65, Part 2, pp 839-856, 1978

88. MARTIN,H.C.,CAREY,G.F., "Introduction to Finite Element Analysis - Theory and Application", McGraw-Hill Book Company, New York, 1973

89. MEEK,J.L., "Matrix Structural Analysis", McGraw-Hill Book Company, New York, 1971

90. MEEK,B.,HEATH,P.,RUSHBY,N., Eds., "Guide to Good Programming Practice", 2nd. edn., Ellis Horwood Limited Publishers, Chichester, 1983

91. MELOSH,R.J., "Structural analysis, fraility evaluation and redesign", Tech. Report No.TR-70-15, Vol.1, Air Force Flight Dynamics Lab., Wright-Patterson Air Force Base, Ohio, July 1970

92. MELOSH,R.J.,JOHNSON,J.R.,LUIK,R., "Survivability analysis of structures", Proc. of the 2nd. Conf. on Matrix Methods in Structural Mechanics, Wright-Patterson Air Force Base, Ohio, AFFDL-TR-68-150, Dec. 1969

93. MELOSH,R.J.,LUIK,R., "Approximate multiple configuration analysis and allocation for least weight structural design",

AFFDL-TR-67-59, Wright-Patterson Air Force Base, Ohio, 1967

94. MELOSH,R.J.,LUIK,R., Closure on "Multiple configuration analysis of structures", J. of the Struct. Div., Proc. of the A.S.C.E., Vol.96, No.ST6, pp 1239-1241, June 1970

95. MELOSH,R.J.,LUIK,R., "Multiple configuration analysis of structures", J. of the Struct. Div., Proc. of the A.S.C.E., Vol.94, No.ST11, pp 2581-2596, Nov. 1968

96. METCALF,M., "Fortran Optimization", Academic Press, London, 1980

97. MICHIELSON,H.F.,DIJK,A., "Structural modifications in redundant structures", J. of the Aero. Sci., Vol.20, No.4, pp 286-288, April 1953

98. MOHRAZ,B.,WRIGHT,R.N., "Solving topologically modified structures", Computers & Structures, Vol.3, pp 341-353, 1973

99. MUXWORTHY,D.T., "Writing Optimizable Fortran", Private communications, March 1979

100. NAYAK,G.C.,ZIENKIEWICZ,O.C., "Elasto-plastic stress analysis. A generalization for various constitutive relations including strain softening", Int. J. for Numerical Methods in Engr., Vol.5, pp 113-135, 1972

101. NAYAK,G.C.,ZIENKIEWICZ,O.C., "Note on the Alpha-constant stiffness method for the analysis of non-linear problems", Int. J. for Numerical Methods in Engr., Vol.4, pp 579-582, 1972

102. NOOR,A.K., "Multiple configuration analysis via mixed method", J. of the Struct. Div., Proc. of the A.S.C.E., Vol.100, No.ST9, pp 1991-1997, Sept. 1974

103. NOOR,A.K., "Recent advances in reduction methods for nonlinear problems", Computers & Structures, Vol.13, pp

31-44, 1981

104. NOOR,A.K., "Survey of computer programs for solution of nonlinear structural and solid mechanics problems", Computers & Structures, Vol.13, pp 425-465, 1981

105. NOOR,A.K.,LOWDER,H.E., "Approximate reanalysis techniques with substructuring", J. of the Struct. Div., Proc. of the A.S.C.E., Vol.101, No.ST8, pp 1687-1698, Aug. 1975

106. NOOR,A.K.,LOWDER,H.E., "Approximate techniques of structural reanalysis", Computers & Structures, Vol.4, pp 801-812, 1974

107. NOOR,A.K.,LOWDER,H.E., "Structural reanalysis via a mixed method", Computers & Structures, Vol.5, pp 9-12, 1975

108. NOOR,A.K.,PETERS,J.M., "Nonlinear analysis via global-local mixed finite element approach", Int. J. for Numerical Methods in Engr., Vol.15, pp 1363-1380, 1980

109. NOOR,A.K.,PETERS,J.M., "Reduced basis technique for nonlinear analysis of structures", AIAA Journal, Vol.18, No.4, pp 455-462, April 1980

110. ODEN,J.T., "Finite Elements of Nonlinear Continua", McGraw-Hill Book Company, New York, 1972

111. ODEN,J.T.,BATHE,K.J., "A commentary on computational mechanics", Applied Mechanics Reviews, Vol.31, No.8, pp 1053-1058, Aug. 1978

112. OWEN,D.R.J.,HINTON,E., "Finite Elements in Plasticity - Theory and Practice", Pineridge Press, Swansea, U.K., 1980

113. PESTEL,E.C.,LECKIE,F.A., "Matrix Methods in Elastomechanics", McGraw-Hill Book Company, New York, 1963

114. PETERSON,R.E., "Stress Concentration Design Factors", John Wiley & Sons, New York, 1953

115. PHANSALKAR,S.R., "Matrix iterative methods for structural reanalysis", Computers & Structures, Vol.4, pp 779-800, 1974

116. PIPES,L.A., "Matrix Methods in Engineering", Prentice-Hall Inc., Englewood Cliffs, N.J., 1963

117. POPPLETON,E.D., "Note on the design of redundant structures", UTIA Tech. Note, No.36, Inst. of Aerophysics, Univ. of Toronto, July 1960

118. POPPLETON,E.D., "The redesign of redundant structures having undesirable stress distributions", J. of the Aerospace Sci., Vol.28, No.5, pp 347-348, April 1961

119. RAIBSTEIN,A.I.,KALEV,I.,PIPANO,A., "Efficient reanalysis of structures by a direct modification procedure", Fifth Nastran User's Colloquim, 1976

120. ROCKEY,K.C.,EVANS,H.R.,GRIFFITHS,D.W.,NETHERCOT,D.A., "The Finite Element Method. A Basic Introduction", Granada Publ. Ltd., London, 1975

121. ROMSTAD,K.M.,HUTCHINSON,J.R.,RUNGE,K.H., "Design parameter variation and structural response", Int. J. for Numerical Methods in Engr., Vol.5, pp 337-349, 1973

122. ROW,D.G.,POWELL,G.H., "A substructure technique for nonlinear static and dynamic analysis", Report No. UCB/EER/C-78/15, Univ. of California, Berkeley, 1978

123. ROW,D.G.,POWELL,G.H.,MONDKAR,D.P., "Solution of progressively changing equilibrium equations for nonlinear structures", Computers & Structures, Vol.7, pp 659-665, 1977

124. SACK,R.L.,CARPENTER,W.C.,HATCH,G.L., "Modification of elements in the displacement method", AIAA Journal, Vol.5, No.9, pp 1708-1710, Sept. 1967

125. SCHMIT,L.A., "Literature review and assessment of the present position", AGARDograph No.149 on Structural Design

and Applications of Mathematical Programming Techniques, pp 34-45, 1971

126. SCHMIT,L.A.,FARSHI,B., "Some approximation concepts for structural synthesis", AIAA Journal, Vol.12, No.5, pp 692-699, May 1974

127. SHERMAN,J.,MORRISON,W.J., "Adjustment of an inverse matrix corresponding to a change in one element of a given matrix", Ann. Math. Statist., Vol.21, pp 124-126, 1950

128. SHERMAN,J.,MORRISON,W.J., "Adjustment of an inverse matrix corresponding to changes in the elements of a given column or a given row of the original matrix", Ann. Math. Statist., Vol.20, pp 621, 1949

129. SOBIESZCZANSKI,J., "Evaluation of algorithms for structrual modification", Proc. of the Conf. on Finite Element Methods in Civil Engr., Vanderbilt Univ., Nashville, Tenn., pp 129-153, 1969

130. SOBIESZCZANSKI,J., "Matrix algorithm for structural modification based upon the parallel element concept", AIAA Journal, Vol.7, No.11, pp 2132-2139, Nov. 1969

131. SOBIESZCZANSKI,J., "Structural modification by perturbation method", J. of the Struct. Div., Proc. of the A.S.C.E., Vol.94, No.ST12, pp 2799-2816, Dec. 1968

132. SPOONER,J.B., "A history of the finite element method", World Congress on Finite Element Methods in Structural Mechanics, Bournemouth, Dorset, England, 12-17 Oct. 1975

133. STORAASLI,O.O.,SOBIESZCZANSKI,J., "Design oriented structural analysis", AIAA Paper 73-338, Williamsburgh, Virginia, 1973

134. STORAASLI,O.O.,SOBIESZCZANSKI,J., "On the accuracy of Taylor approximation", AIAA Journal, Vol.12, No.2, pp 231-233, Feb. 1974

135. STRICKLIN,J.A.,HAISLER,W.E., "Survey of solution procedures

for nonlinear static and dynamic analysis", SAE Intern. Conf. Vechicle Structural Mechanics, Detroit, Michagan, March 1974

136. TILLERSON,J.R.,STRICKLIN,J.A.,HAISLER,W.E., "Numerical methods for the solution of nonlinear problems in structural analysis", ASME Winter Annual Meeting, Detroit, Nov. 1973, Numerical Solution of Nonlinear Structural Problem (Ed. HARTUNG,R.F.), AMD Vol.6, pp 67-101, 1973

137. TOPPING,B.H.V., "The Application of Dynamic Relaxation to the Design of Modular Space Structures", A thesis submitted for the Degree of Doctor of Philosophy, The City University, London, 1978

138. TOPPING,B.H.V., "The application of the theorems of structural variation to finite element problems", Int. J. for Numerical Methods in Engr., Vol.19, pp 141-144, 1983

139. TOPPING,B.H.V.,MAJID,K.I.,CHAN,H.F., "The theorems of geometric variation", to be published

140. TOTTENHAM,H.,BREBBIA,C., Eds., "Finite Element Techniques in Structural Mechanics", Stress Analysis Publ., Southampton, England, 1970

141. TURNER,M.J.,CLOUGH,R.W.,MARTIN,H.C.,TOPP,L.J., "Stiffness and deflection analysis of complex structures", J. of the Aero. Sci., Vol.23, No.9, pp 805-823, 854, Sept. 1956

142. WANG,B.P.,PILKEY,W.D., "Efficient reanalysis of locally modified structures", Proc. 1st. Chautaqua on Finite Element Modelling, pp 37-61, Schaeffer Analysis, 1980

143. WANG,B.P.,PILKEY,W.D., "Efficient reanalysis of locally modified structures", Dept. of Mech. & Aerospace Engr., Charlottesville, Virginia Univ., 1980

144. WANG,B.P.,PILKEY,W.D., "Parameterization in finite element

analysis", Proc. Int. Symp. on Optimum Structural Design, Tuscon, Arizona, pp 7.1-7.7, 1981

145. WANG,B.P.,PILKEY,W.D.,PALAZZOLO,A.R., "Reanalysis, modal synthesis and dynamic design", Chapter 8 of State-of-the-Art Surveys on Finite Element Technology, (Eds. NOOR,A.K.,PILKEY,W.D.), The American Society of Mechanical Engineers, New York, 1983

146. WASHIZU,K., "Variational Methods in Elasticity and Plasticity", 1st. edn., Pergamon Press, Oxford, U.K., 1968

147. WEIL,N.A.,NEWMARK,N.M., "Large deflections of elliptical plates", J. Appl. Mech., Vol.23, pp 21-26, March 1956

148. WEINER,B.L., "Variation of coefficients of simultaneous linear equations", Trans. A.S.C.E., Vol.113, pp 1349-1390, 1948

149. WIBERG,N.E., Discussion of "Efficient reanalysis of modified structures", J. of the Struct. Div., Proc. of the A.S.C.E., Vol.97, No.ST10, pp 2612-2619, Oct. 1971

150. WOOD,R.D., "The Application of Finite Element Methods to Geometrically Non-linear Finite Element Analysis", A thesis submitted for the Degree of Doctor of Philosophy, Univ. of Wales, Swansea, 1973

151. YANG,W.H., "A method for updating Cholesky factorization of a band matrix", Computer Methods in Applied Mechanics & Engr., Vol.12, pp 281-288, 1977

152. YOUNG,R.C., "Efficient nonlinear analysis of factored matrix modification", Trans. of the 4th. Int. Conf. on Struct. Mech. in Reactor Tech.,(Eds. JAEGER,T.A.,BOLEY,B.A.), San Francisco, Calif., pp M4/3, Aug. 1977

153. ZIELKE,G., "Inversion of modified symmetric matrices", J. Assoc. Comp. Mach., Vol.15, pp 402-408, 1968

154. ZIENKIEWICZ,O.C., "The Finite Element Method", 3rd. edn.,

McGraw-Hill Book Company, U.K., 1977

155. ZIENKIEWICZ,O.C., "The finite element method : From intuition to generality", Applied Mechanics Reviews, Vol.23, No.3, pp 249-256, March 1970

156. ZIENKIEWICZ,O.C., "The generalized finite element method - State of the art and future directions", J. Appl. Mech., Vol.50, pp 1210-1217, Dec. 1983

157. ZIENKIEWICZ,O.C.,NAYAK,G.C., "A general approach to problems of plasticity and large deformation using isoparametric element", Proc. of the Conf. on Matrix Methods in Struct. Mech., Wright-Patterson Air Force Base, Ohio, AFFDL-TR-71-160, Dec. 1973

158. ZIENKIEWICZ,O.C.,OWEN,D.R.J.,PHILLIPS,D.V.,NAYAK,G.C., "Finite element methods in the analysis of reactor vessels", Nuclear Engr. & Design, Vol.20, pp 507-541, 1972

159. ZIENKIEWICZ,O.C.,VALLIAPPAN,S.,KING,I.P., "Elasto-plastic solutions of engineering problems 'initial stress', finite element approach", Int. J. for Numerical Methods in Engr., Vol.1, pp 75-100, 1969

160. ZIMMERMANN,K.J,SPENCE,R., "Interactive use of finite element programs - or - How to get more out of your finite element model", Computers & Structures, Vol.12, pp 633-638, 1980

161. ABU KASSIM,A.M.,TOPPING,B.H.V., "Static reanalysis of structures: A review", Civil Comp 85, The Proc. of the 2nd. Int. Conf. on Civil and Struct. Engr. Comp., Civil Comp. Press, Edinburgh, 1985